NAVAL POSTGRADUATE SCHOOL
MONTEREY, CALIFORNIA

THESIS

PROPOGATION OF FIRE GENERATED SMOKE IN SHIPBOARD SPACES

by

Michael D. Mehls

March 2000

Thesis Advisor: M. D. Kelleher

Approved for public release; distribution is unlimited.

20000612 010

REPORT DOCUMENTATION PAGE

Form Approved OMB No. 0704-0188

1. AGENCY USE ONLY *(Leave blank)*	2. REPORT DATE March 2000.	3. REPORT TYPE AND DATES COVERED Master's Thesis

4. TITLE AND SUBTITLE: Propagation of Fire Generated Smoke in Shipboard Spaces.	5. FUNDING NUMBERS
6. AUTHOR(S) Mehls, Michael D.	

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) Naval Postgraduate School Monterey CA 93943-5000	8. PERFORMING ORGANIZATION REPORT NUMBER
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)	10. SPONSORING/MONITORING AGENCY REPORT NUMBER

11. SUPPLEMENTARY NOTES
The views expressed here are those of the authors and do not reflect the official policy or position of the Department of Defense or the U.S. Government.

12a. DISTRIBUTION/AVAILABILITY STATEMENT Approved for public release; distribution is unlimited.	12b. DISTRIBUTION CODE

13.   ABSTRACT *(maximum 200 words)*
The propagation of fire generated smoke into a shipboard space has been computationally modeled using a commercial code generated by Computational Fluid Dynamics Research Corporation (CFDRC). This study was based on space 01-163-2-L of an Arleigh Burke Class Flight IIA Destroyer. However, with changes, the model can be reconfigured to represent other shipboard spaces. Multiple smoke scenarios are applied to the space. For all scenarios, the inlet used is forward water tight door. Smoke enters the upper half of the door, while air enters through the bottom half. The temperature of the inlet fluids is altered to observe its effect on propagation. In the last scenario, the floor temperature is isothermally held at 1200 K to simulate a fire in the space below. The results of this scenario shows that extreme temperatures of adjacent spaces has minimal effect on propagation. The overall goal of this study is to show how computational methods can be used to model propagation of smoke in shipboard spaces.

14. SUBJECT TERMS Convection, Smoke Modeling, Computational Fluid Dynamics	15. NUMBER OF PAGES 75
	16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT Unclassified	18. SECURITY CLASSIFICATION OF THIS PAGE Unclassified	19. SECURITY CLASSIFICATION OF ABSTRACT Unclassified	20. LIMITATION OF ABSTRACT UL

NSN 7540-01-280-5500

Standard Form 298 (Rev. 2-89)
Prescribed by ANSI Std. 239-18 298-102

PROPAGATION OF FIRE GENERATED SMOKE IN SHIPBOARD SPACES

Michael D. Mehls
Lieutenant, United States Navy
B.S., United States Naval Academy, 1992

Submitted in partial fulfillment of the
requirements for the degree of

MASTER OF SCIENCE IN MECHANICAL ENGINEERING

from the

NAVAL POSTGRADUATE SCHOOL
March 2000

Author: _____

Approved by: _____
Matthew D. Kelleher, Thesis Advisor

_____
Terry R. McNelley, Chairman
Department of Mechanical Engineering

ABSTRACT

The propagation of fire generated smoke into a shipboard space has been computationally modeled using a commercial code generated by Computational Fluid Dynamics Research Corporation (CFDRC). This study was based on space 01-163-2-L of an Arleigh Burke Class Flight IIA Destroyer. However, with changes, the model can be reconfigured to represent other shipboard spaces. Multiple smoke scenarios are applied to the space. For all scenarios, the inlet used is forward water tight door. Smoke enters the upper half of the door, while air enters through the bottom half. The temperature of the inlet fluids is altered to observe its effect on propagation. In the last scenario, the floor temperature is isothermally held at 1200 K to simulate a fire in the space below. The results of this scenario shows that extreme temperatures of adjacent spaces has minimal effect on propagation. The overall goal of this study is to show how computational methods can be used to model propagation of smoke in shipboard spaces.

TABLE OF CONTENTS

ACKNOWLEDGEMENT

I would like to thank Professor Kelleher for his patience and guidance during this thesis. Additionally, I would like to thank Mark Rist and Christine Frieder of CFDRC for their software support. Most of all, I would like to thank my wife, Elise, and daughter, Rachael, for their unending support and encouragement. Finally, I would like to close with a verse, because none of this would have been possible without the blessings bestowed upon us by the Lord.

Consider it pure joy, my brothers, whenever you face trials of many kinds, because you know that the testing of your faith develops perseverance. Perseverance must finish its work so that you may be mature and complete, not lacking anything. If any of you lacks wisdom, he should ask God, who gives generously to all without finding fault, and it will be given to him.   James 1: 2-5

I. INTRODUCTION

A.   BACKGROUND

In 1987, USS STARK (FFG-31) was struck by two Iraqi Exocet missiles while on patrol in the Arabian Gulf. Even though the missiles failed to detonate, the crew had a variety of obstacles to overcome in order to save the ship. The largest problem facing them was the huge amount of unburned fuel that was producing enormously high temperatures, and causing thermal damage to the ship's mid-section. See Figure 1. Decks and electrical cableways, which melt at 2800°F (1811°K), were no match for the 3000° F(1922° K) produced by the unburned fuel. In addition to the tremendous amount of heat, the crew was forced to fight the blaze through an enormous amount of smoke. This hampered the crew's ability to even get near the fight the fire and challenged the Navy's fire fighting technology.

> The dense smoke severely limited the men's ability to investigate spaces and to maintain sufficient breathing support. Early on, smoke completely engulfed the main-deck passageway, and many below-deck spaces had to be abandoned because of smoke before the impact of heat and flames was felt. Contributing to this difficulty was the toxicity of the smoke produced by the burning PVC-jacketed electrical cables and antisweat pipe insulation installed throughout the ship. [Ref 1]

While the USS STARK had a state of the art fire protection system, it proved to be no match for the extreme temperatures of a "weapon induced" fire. This extremely high temperature fire is the greatest danger to any ship, even the Navy's newest Arleigh Burke class destroyer.

THIS PAGE LEFT INTENTIONALLY BLANK

BRIDGE

WARDROOM

GALLEY

CREW BERTHING

MK 13
MISSILE
MAG.
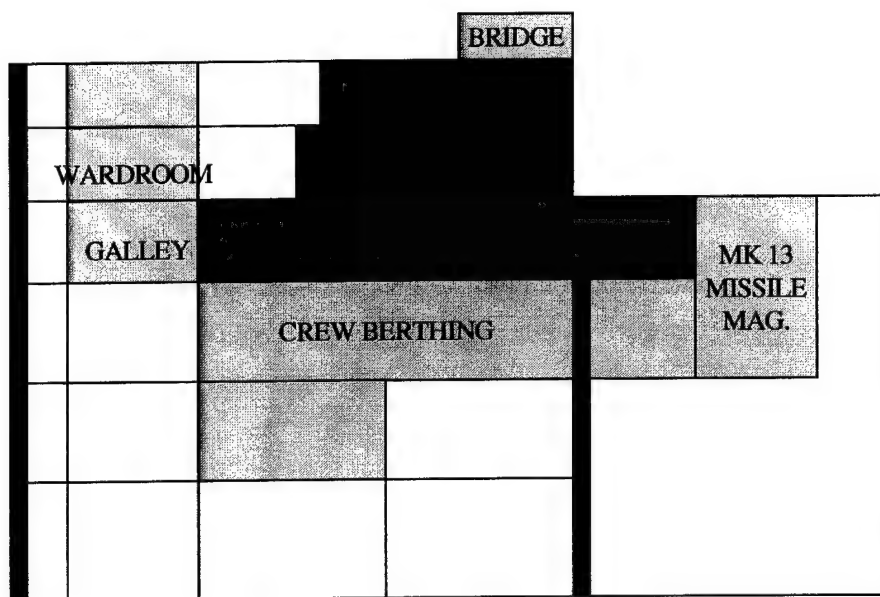
■ FIRE BOUNDARY

▨ SMOKE/HEAT DAMAGE

■ FIRE DAMAGE

Figure 1. USS STARK (FFG-31). View of Starboard Side Damage

THIS PAGE INTENTIONALLY LEFT BLANK

B. DESCRIPTION OF THE PROBLEM

The engineering profession has realized that it is more cost effective to model and test a computer generated design than to build and test the same design in a laboratory. With the recent advances in desktop computers, engineers are now able to accurately model three-dimensional heat transfer and fluid flow problems. These advances could be used to design future ships by modeling shipboard spaces in order to determine the hazards of smoke resident in each design. If problems are discovered, improvements to the design could be made and then tests could be computed again to determine if the flaw still exists. This could save both time and money!

This strategy could and should be used for fire protection systems resident in shipboard spaces. Fire protection engineers have not drastically changed the way in which ships are designed since the Perry Class was commissioned. With the speed and memory capability of the latest generation of desktop computers, problems can be solved, while they are still in a computationally generated form.

Since the Surface Combatant of the 21st Century (DD-21) is being designed for a maximum of 95 personnel, an enormous amount of planning will be required in order to ensure that the ship's personnel will be able to safely defend the ship against fire and smoke. After all, fighting a fire is the most manpower intensive evolution that U.S. Navy ships undergo. The results of this study will give engineers another tool to determine realistic scenarios, which can be used to support or reject the proposed manning requirements.

C.   PREVIOUS WORK

Kay and Morris [Ref 2] examined the development of smoke modeling. They determined that fire safety aboard naval vessels was based on engineering judgement and previous fire incidents. However, with the speed of modern processors, computer modeling is possible, which provides engineers with a range of possibilities for solving the same problem. One can compare and contrast these solutions using either zone modeling or field modeling in the form of computational fluid dynamics (CFD). Using CFD there are two approaches to take: deterministic or probabilistic modeling. As expected, the probabilistic model quantifies the probability of fire spread. While these results may satisfactorily provide a quick answer in accordance with the laws of probability, shipboard fires spread based on the laws of nature, which may or may not coincide with the laws of probability. A better approach is to develop a physically based deterministic model, which produces more realistic information(temperature, velocity, pressure etc). This method can better be assimilated to real world scenarios providing engineers with a broader approach, which is better suited to solve fire safety problems.

In 1996, Tabarra, Kenrick and Matthews [Ref 3] analyzed natural smoke movement in a model tunnel using a CFD validation. They found "that a strong ventilation system may not be the best answer to passenger safety, as this tends to mix the hot smoke layers with the cool fresh air drawn in by the fire." Their nemesis was the relatively low computational power of computers, but their coarse models accurately represented the flow. Unfortunately, radiation and turbulence were neglected, because their grids were already maximized to the computers' limits.

Chow [Ref 4] used a CFD package, entitled PHOENICS, to computationally model a fire inside a building and how the resulting stratified smoke layer changed with different forced ventilation schemes. Also, using his models, he was able to predict the temperature of a well-mixed model if hot smoke is mixed with cool air.

Null [Ref 5] used CFD-ACE to model the Concentric Canister Launcher(CCL) when it is exposed to a fire in an adjacent space aboard Navy ships. His research was valuable to predicting the temperatures inside the CCL, because once a cook-off temperature is reached, the energetic material (solid propellant, liquid fuels and explosives) in the missile could explode. The impetus behind his research was the unburned fuel of the Exocet missiles which hit the USS STARK(FFG-31) in 1987. The unburned fuel produces fires in excess of 3000° F (1922° K)

To date, CFD research has focused on two-dimensional analysis, but with the recent advances in computational speeds and memory capacities, three-dimensional analysis is becoming feasible for researchers with just a desktop personal computer. Blay, Tuhault and Joubert [Ref 6] investigated a free convection problem in a corridor with smoke being removed through the ceiling via a soffit. The box in the lower left corner was a heat generating source, which was the driving force behind free convection. Their results included isothermal lines and velocity vectors of the smoke movement in the corridor. Although not stated, it is assumed that the CFD code, used to model the flow, was generated by the authors.

D. OBJECTIVES

The purpose of this study is to develop and examine a computationally generated model which will give engineers a better understanding of how smoke travels in

shipboard spaces. While the model will be generated in the form of passageway 01-163-2-L, onboard an Arleigh Burke (DDG-51) class destroyer, additional inlets and outlets for the smoke have been added to allow more opportunities to model other spaces or other scenarios. Furthermore, the source of the fire will determine the physical properties associated with the smoke, such as density, viscosity etc. These properties will have a dramatic effect on the movement of the smoke.

II. COMPUTATIONAL FLUID DYNAMICS

A. OVERVIEW

The software package used to analyze the smoke propagation was developed by CFD Research Corporation (CFDRC). It is entitled CFD-ACE [Refs 7-10] and uses three separate, yet interactive codes to solve the fluid flow problem. See Figure 2. The codes are GEOM, GUI and VIEW. CFD-GEOM is the pre-processor, which offers comprehensive geometry and grid generation, enabling the user to generate a grid representing the architecture of the problem. This stage will consume approximately 90 percent of the total input time, due to the large number of inputs. CFD-GUI allows the user to input the specific thermophysical properties associated with the fluids. This includes the initial conditions (density, velocity), boundary conditions (adiabatic, isothermal) and interaction of species within the problem (heat transfer, turbulence, mixing). The user designates the number of iterations. Likewise, the results can be slightly varied by altering the differencing scheme, which will be explained in more detail later. Altering the relaxation method or amount of constraint can change the solution to the conservation equations. After all of the parameters have been entered, CFD-GUI solves the series of equations. The last stage, CFD-VIEW, is the postprocessor, which displays the results graphically through colored gradients, vectors or animation.

B. FINITE VOLUME ANALYSIS

CFD-ACE divides the geometry into a finite number of volumes, each generated by a structured grid that uses single i, j, k coordinates to identify a particular direction within the finite volume. Various coordinate systems can be used in order to represent the problem ie. cartesian, cylindrical or spherical. Shipboard spaces are primarily rectangular, thus, a cartesian system was chosen.

The following was taken from the Theory Manual of CFD-ACE and best describes the theory in the finite volume approach:

This means that the discretized equations are formulated by evaluating and integrating fluxes across the faces of control volumes in order to satisfy the relevant conservation equations. Each control volume is, topologically, a cube with six faces and six direct neighbors. The cell centered aspect of the CFD-ACE approach means that each dependent variable is solved for at the center of each control volume and the values obtained are considered to prevail over the entire control volume. When evaluating convective fluxes over a control volume, differencing schemes of varying accuracy can be used. CFD-ACE offers a range of differencing schemes from first-order upwind, to second-order central differencing, to third-order schemes in high-speed flows. These schemes can be independently selected for each variable to be solved.

Using a finite volume approach, the governing equations can be discretized into cells. CFD-ACE uses a cell centered variable arrangement where all flow variables and fluid properties are stored at the center of each cell. The following equation is derived for each variable in each finite volume:

$$a_p \phi_p = a_e \phi_e + a_w \phi_w + a_n \phi_n + a_s \phi_s + a_h \phi_h + a_l \phi_l + S$$

The meaning of the symbols is as follows: ϕ is the flow quantity, such as temperature, velocity or concentration. It is composed of average and instantaneous components. P refers to the center of the cell and e(East), w(West), n(North), s(South), h(high) and l(low) represent the values at the six adjacent faces of that cell, as shown in Figure 3.
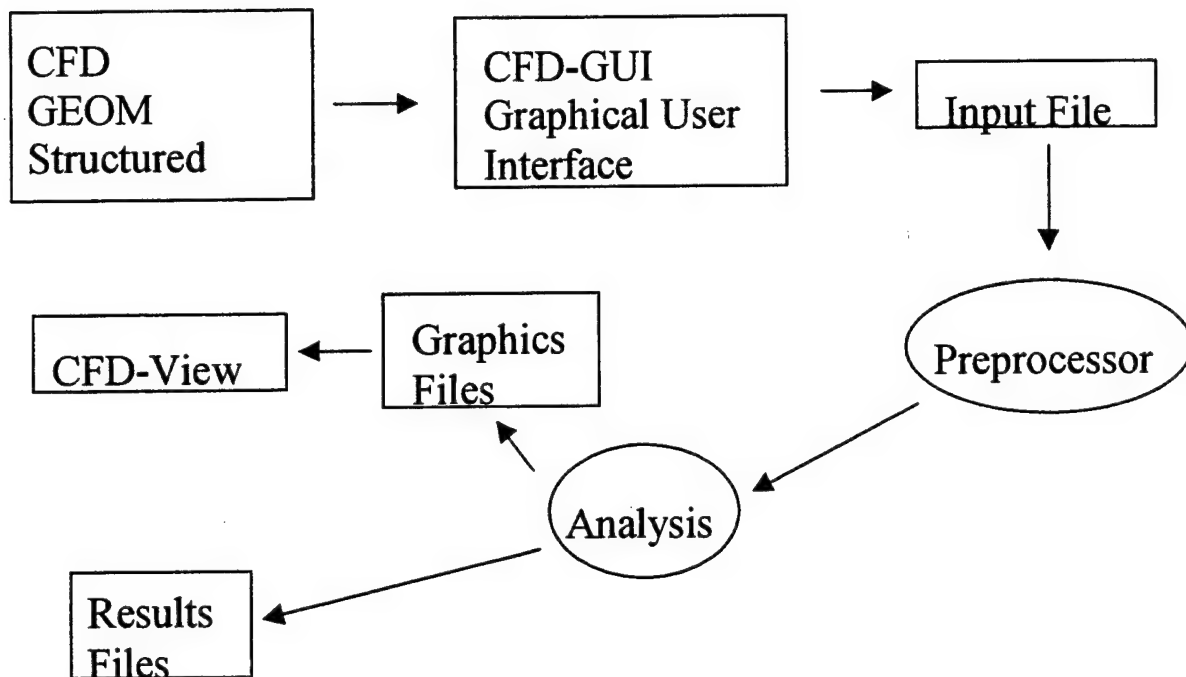
Figure 2. Schematic Representation of CFD-ACE
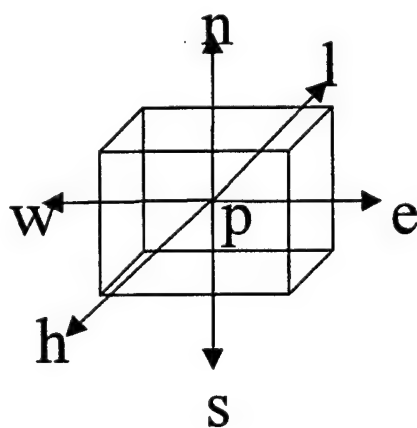

Figure 3. Six Directions of Finite Volume Cube

The source term, S, represents terms other than the convection and diffusion effects, such as the pressure gradient. The coefficient 'a' represents the effects of convection and diffusion across the cell faces, which can be represented as follows:

Convection Term $\quad -\rho_e A_e u_e \phi_e \quad$ Diffusion Term $\quad -\Gamma_e \dfrac{(\phi_p - \phi_e)}{\Delta_e} A_e$

A_e is the area across the east face and Δ_e is the distance from the center of the cell to the East face. ρ represents density, while Γ is the molecular diffusion coefficient of the fluid.

The dynamics of this fluid problem are driven by conduction, convection, mass diffusion and radiation. In order to understand the problem, the governing equations must be analyzed. Only the most general version of the equations will be used:

The Continuity Equation for compressible fluid is: $\dfrac{\partial \rho}{\partial t} + div(\rho u) = 0$

The Continuity Equation for incompressible fluid is: $\dfrac{\partial u}{\partial x} + \dfrac{\partial v}{\partial y} + \dfrac{\partial w}{\partial z} = 0$

In the Momentum Equation, p is the static pressure, τ_{ij} is the viscous stress tensor and f_i is the body force.

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_i}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_i} + \rho f_i$$

For Newtonian fluids, τ_{ij} can be related to the velocity gradients through

$$\tau_{ij} = \mu(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}) - \frac{2}{3}\mu(\frac{\partial u_k}{\partial x_k})\delta_{ij}$$

where μ is the fluid dynamic viscosity and δ_{ij} is the Kronecker delta. Substitution of the above two equations results in the Navier-Stokes Equation:

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}(\mu\frac{\partial u_i}{\partial x_i} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\mu\frac{\partial u_k}{\partial x_k}\delta_{ij}) + \rho f_i$$

In the Energy Equation, shown below, J_{ij} is the total (concentration-driven and temperature driven) diffusive mass flux for species i. Also, h_i represents the enthalpy for species i, S_a stands for additional sources due to surface reaction, radiation and liquid spray and q_j is the j-component of the heat flux.

$$(\rho h) + \frac{\partial}{\partial x_j}(\rho u_j h) = -\frac{\partial q_j}{\partial x_j} + \frac{\partial p}{\partial t} + u_j\frac{\partial p}{\partial x_j} + \tau_{ij}\frac{\partial u_i}{\partial x_j} - \frac{\partial}{\partial x_i}(J_{ij}h_i) + S_a$$

Fourier's Law is employed to model the conduction heat flux where K is the thermal conductivity.

$$q_j = -K\frac{\partial T}{\partial x_j}$$

CFD-ACE employs an iterative solution scheme in which the assembled equations for each dependent variable are solved sequentially and repeatedly with the goal of improving the solution with each iteration. This reduces the errors to acceptably small values. The nonlinear and coupled nature of the equations makes it necessary to restrain or under-relax the iteration-to iteration changes in the variable in order to obtain convergence of the solution procedure.

The differencing scheme determines how the cell face values are calculated. The differencing scheme used in this model is the first-order upwind scheme. This means that the value of ϕ at any given point is dependent upon the value of ϕ at the adjacent point, just upstream of the flow.

13

For each iteration, the pressure (P) and velocity (U) terms will equal the current value plus some unknown correction according to the following equation:

$$P = p^* + p'$$

$$U = u^* + u'$$

where P represents the current value of pressure, while p* represents the previous value of pressure as P is undergoing the iterative process and p' is the unknown correction that exists between iterations. The symbols and process for computing the velocity remain the same as for the pressure.

When the correction approaches zero then we can be reasonably confident that the value of pressure and velocity are accurate for the given flow field. The values of these corrections are determined by using the momentum equation to develop a functional relationship between the pressure and velocity fields. The pressures are then substituted for the velocities into a discretized continuity equation and the simultaneous matrix of equations is solved over the entire region. The solution procedure is summarized in the figure below. Figure 4 shows that for each iteration, within a transient time step, a system of equations must be solved for each dependent variable. A benefit to using the structured grid geometry is that it produces a banded matrix of coefficients for each dependent variable. The solution methods used for each system of equations are Backward Euler and a version of Forward Differencing Technique.

In order to ensure that the solution for each iteration does not diverge, under-relaxation must be applied to the dependent and auxiliary variables. Under-relaxation constrains the amount that each variable can change from one iteration to the next. The

dependent variables (u,v,w,k) are modified using an Inertial Factor. A linear under-

relaxation technique is applied to the auxiliary variable(p,T,μ, ρ).
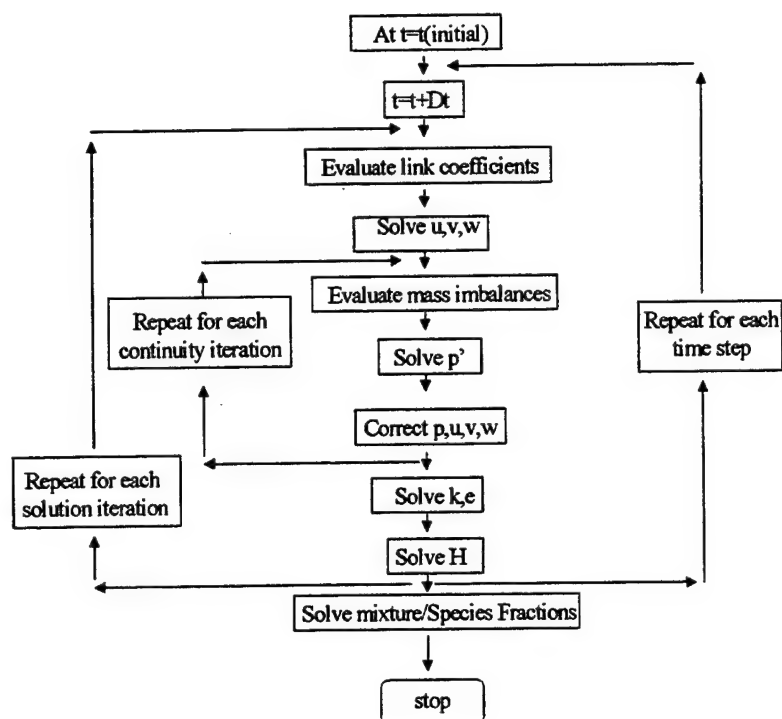

Figure 4. CFD-ACE Flow Solver Solution Procedure[Ref 7].

THIS PAGE INTENTIONALLY LEFT BLANK

III. MODEL

A. GEOMETRY

1. Model Selection

The parameters for this model were taken from passageway 01-163-2-L of an Arleigh Burke Class Destroyer (DDG-21). The dimensions of the model identically match the actual passageway. Figure 5 shows a plan view of the compartment as shown in the ship's drawings. This passageway (p-way) was chosen, because it is geometrically simple and has a variety of openings for smoke intrusion. The model allows a variety of smoke propagation scenarios to be studied. For example, even though the model may have an entrance in a specific location, unless that door is activated as an inlet or outlet, the program will treat it as a wall. Therefore, the non-activated entrances and exits will have no effect on the results. For example, this model could be used to represent the manner in which smoke enters a space, either horizontally or vertically. The model could also could be used to determine the rate or path which the smoke exits the space. This could be accomplished through a desmoking procedure using either forced or natural means, or it could be used to determine the affects of natural circulation in the skullery.

2. Grid Distribution and Model Generation

The simulations were carried out using a Micron Client Pro Desktop computer, with 128 megabytes of Ram and an internal 12 gigabyte hard drive. The software used was CFD-ACE version 5.0, which was last updated in February, 1999.

Figure 5. Plan View of Space 01-163-2-L

Generating a finite volume mesh is a sequential process. The first step is to generate a skeletal model as shown in Figure 6. This rough sketch shows the large rectangular outer structure of the space and the location of the hatches.

The next step in generating a finite volume is to superimpose a grid upon each of the lines in Figure 6. Each major edge was allowed a total of 30 grid points. This meant that each of the smaller edges needed to be edged proportionally, according to its length. This formed an intricate puzzle, continuously summing the edges to ensure the totals of opposite edges matched. For most edges, this was not difficult due to the symmetrical nature of the planes perpendicular to the y and z axes. However, juggling the edge points of adjacent walls was difficult.

Once the edging process was complete, faces were generated by designating four connected, edges. In most cases, rectangles resulted from connecting the four edges. If desired, any of the faces could be designated as an inlet or an outlet. This provides a great deal of the flexibility in the model. There are a total of 119 faces, as shown in Figure 7. This model allows the investigation of a variety of scenarios. Circles are used to represent scuttles (round, vertical entrances or exits to a shipboard space). However, circles need to be generated in a series of four arcs, which must follow the same restrictions as the four sides of a rectangle. These four arcs are again grouped in order to form a face.

Once all of the faces have been generated, a block is formed. A block can best be defined as a group of six adjacent faces. For example, the rectangular volume, shown in Figure 8, has sides located at X=Y=Z=0 and an opposite face at their respective maximum values.
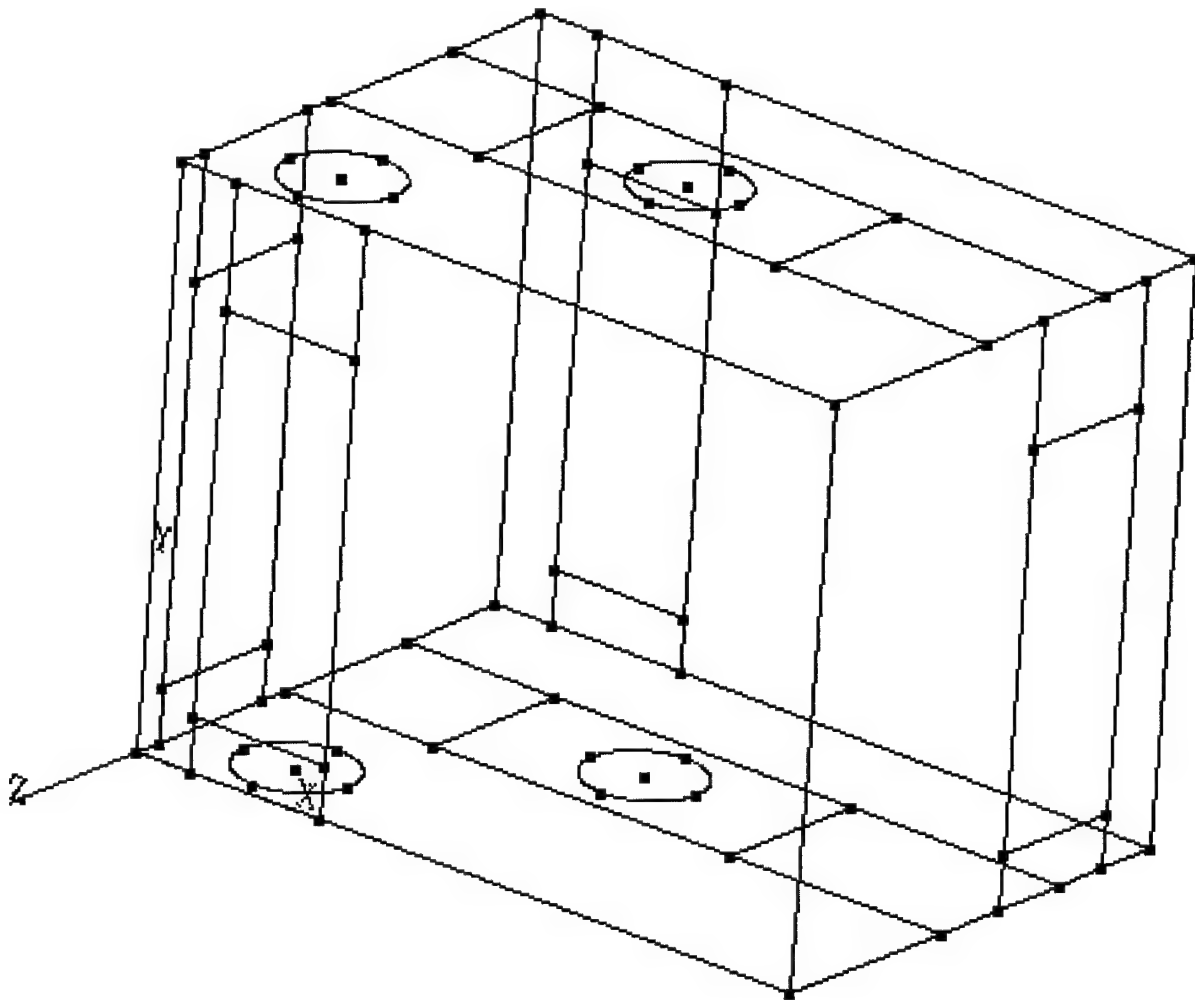
THIS PAGE INTENTIONALLY LEFT BLANK

Figure 6. Line Drawing of Model using CFD-ACE
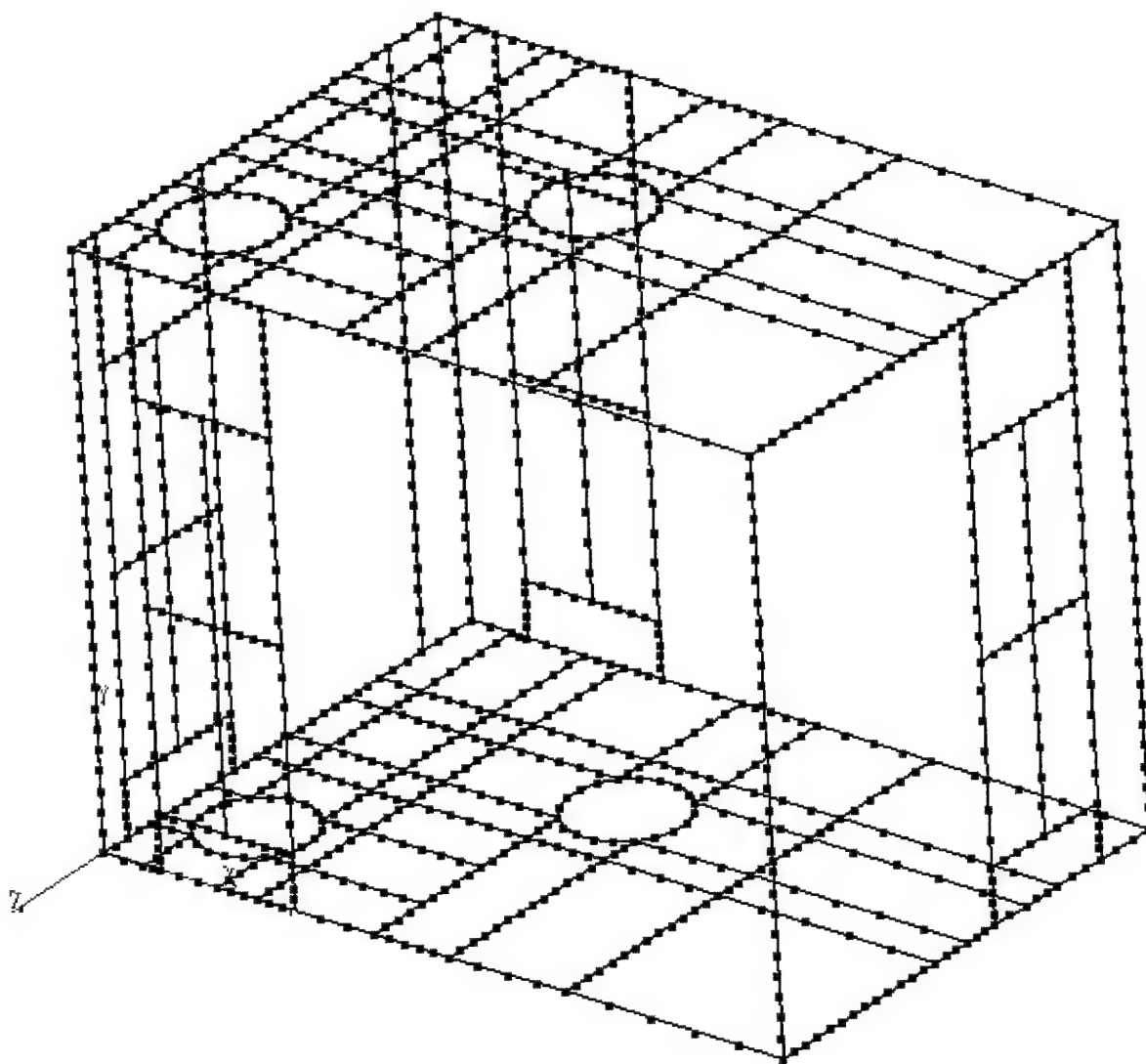
THIS PAGE INTENTIONALLY LEFT BLANK

**Figure 7. CFD Representation of P-WAY 01-163-2-L of DDG-51 Class (Extra
hatches added to increase the number of possible scenarios)**
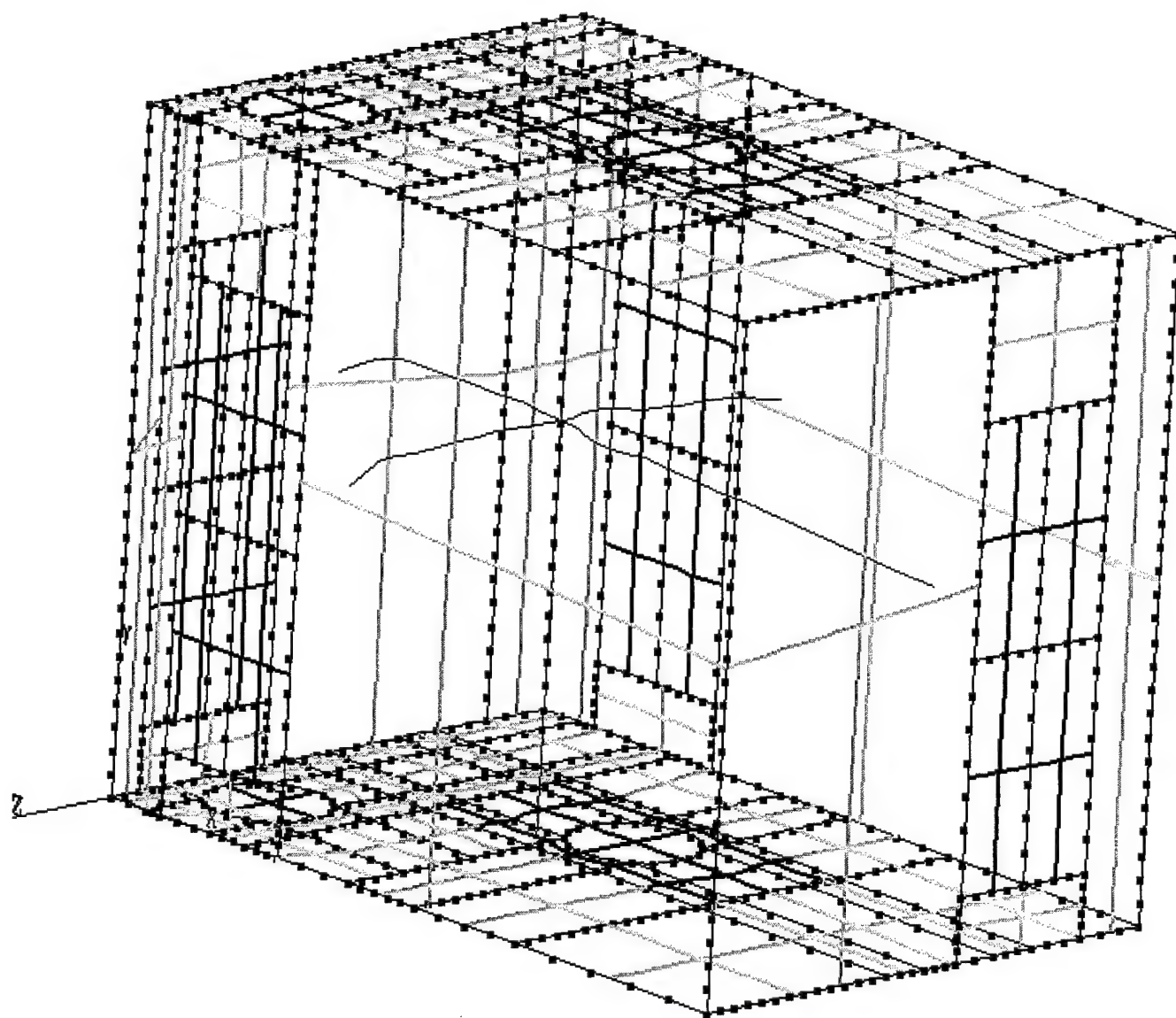
THIS PAGE INTENTIONALLY LEFT BLANK

Figure 8. The Black Lines designate the Inlets and Outlets of the compartment.

THIS PAGE INTENTIONALLY LEFT BLANK

Once all six sides are designated, the code will recognize this new entity as a completely closed structure, called a domain. If desired, multiple domains could be linked to model a series of shipboard spaces.

B. THERMOPHYSICAL MODEL

After the physical dimensions have been constructed in CFD-GEOM, the domain is transferred into the numerical solution generation portion of CFD-ACE, entitled, CFD-GUI. The type of fluid flow problem is then determined: incompressible flow, heat transfer, radiation etc. Based on the type of problem entered, GUI requires inputs (initial conditions, boundary conditions) in order to solve the conservation equations. Many of the inputs are constant, such as the molecular weight. For, example, air is 29 g/mol and smoke is modeled as a gas with a molecular weight of 46 g/mol. However, others properties will change as the problem evolves. For example, the gaseous viscosity is computed by using Sutherland's Law as shown below:

$$\mu = \frac{AT^{1.5}}{B+T}$$

A and B are determined by the gases and T represents the temperature as it changes during the iterative process. The Joint Army, Navy, NASA, and Air Force (JANNAF) Tables [Ref 7] are used to estimate the specific heat, enthalpy and entropy for a gaseous species. When the Ideal Gas Law is selected, density is calculated as shown below:

$$\rho = \frac{(p+P_{ref})M}{RT}$$

P_{ref} is the reference pressure, M is the fluid molecular weight and R is the universal gas constant. T and p are the locally calculated values of relative temperature and pressure respectively.

Turbulence is modeled using the standard k-ε model. It is a two equation model that employs partial differential equations to govern the transport of turbulent kinetic energy, k, and its dissipation rate, ε. CFD-GUI utilizes the Launder and Spalding model (1974). Further details can be found in Ref 7.

Spatial differencing allows the governing equations to be discretized, so the advantages of the finite volume method can be utilized. The finite volume method recognizes that the any property belonging to a finite volume will be influenced by all six adjacent cells. However, the upwind differencing scheme takes this idea one step further. It adds a weighting factor to the properties associated with each of the adjacent cells. The greatest weight goes to the cell, which is directly upwind of a given cell. Likewise, reduced weights are assigned to cells, which are downwind of a given cell, because their ability to change the given cell will be reduced due to the flowing stream. All scenarios used the upwind differencing scheme.

C. REQUIRED INPUTS

Due to the complex nature of CFD terminology, a brief description of the required inputs seems appropriate.

1. Under-relaxation

Under-relaxation is used to constrain the change in the variable from iteration to iteration in order to prevent divergence of the solution procedure. Large values of under-relaxation are associated with variables that have an increased risk of becoming divergent.

2. Sweeps

Sweeps can best be described as a local iteration within the global iteration process. As the global finite volume process iterates, the solver will iterate a given cell an increased number of times in order to prevent the solution from diverging. For example, the default setting for all velocity sweeps is 5. However, pressure can cause a solution to diverge quicker than other variables, therefore it's default setting is 30.

3. Initial and Boundary Conditions

The grid is based on a cartesian coordinate system with U, V, and W velocities aligning themselves with the X, Y, and Z axes, respectively. All pressures in CFD-ACE are relative to a reference pressure, which is set at 100000 N/m^2. Qualitatively, Turbulence Kinetic energy (K) is total amount of energy associated with an eddy. Just as the title indicates, Turbulence Dissipation Rate (D) is the rate at which the eddy loses energy. Turbulent Length Scale (L) is estimated at approximately 0.3% of the size of the inlet. Only two of the previous three variables are required to process the solution, while the third is computed using the equations below:

$$D = \frac{11.74K^{1.5}}{L}$$

If all three variables are entered, L is ignored and generated via the above equation, using D and K.

THIS PAGE INTENTIONALLY LEFT BLANK

IV. RESULTS

The main objective of this research was to use a commercially-developed CFD program to generate a three-dimensional model of the smoke dissipation within the configuration of a shipboard space.

With the assumption that this is the first research to be done coupling this CFD software with the goal of shipboard space modeling, the accuracy of the results with respect to a live testing is unclear. However, one internal method of measuring the accuracy is to view the residual plots. For each scenario completed, the residuals all decreased at least four orders of magnitude, which meets CFDRC's criteria for convergence. Therefore, the results can be considered valid.

Based on the success in the residual converging for each scenario, it leads this engineer to believe that the CFD code has not been pushed to its limits. Most scenarios in fluid flow require alterations of the solution monitors (sweeps or under-relaxation) in order for a solution to converge. This study regularly used the recommended values for sweeps and under-relaxation and the solution still converged. This indicates that a more violent flow scenario (higher velocities or increased turbulence) could have been modeled before the solution failed to converge. Of course, this would then require the inherent monitors to be altered to regain a converging solution.

As with any modeling software, the results are only as good as the input data. CFD-ACE is no exception. The user needs to be aware that the slightest change in any input data can drastically change the output. Other dramatic changes can be viewed by changing the iterative solving mechanisms, such as the relaxation or sweep parameters.

When contrasted, scenarios A, B and C exhibit small differences in the diffusion rates. This is attributable due to the greater molecular mass of the smoke, 46 g/mol, compared to the air, at only 30 g/mol.

Contrasting Scenarios A and C, it is apparent that the dissipation rate is not affected, even when the floor is exposed to such extreme temperatures.

V. CONCLUSIONS

This study's results successfully modeled smoke propagation in shipboard spaces. The accuracy of the results is valid as verified through the use of residuals.

This study successfully pioneered the use of today's CFD technology to simulate the effects of smoke dissipation in shipboard spaces. These models were generated in order to develop a greater appreciation for the numerous forces that alter smoke movement. As the power of computer and increased accuracy of CFD technology grow together, this method of shipboard simulation will one day become the norm for system design and engineering.

THIS PAGE INTENTIONALLY LEFT BLANK

VI. RECOMMENDATIONS

The following recommendations are made in the continuation of this study:

- Run the model using a fixed pressure at the inlet and the outlet. The Arleigh Burke Class has a unique CBR pressurizing system, which will most likely be the driving force behind smoke propagation.

- Change the geometry to reflect the presence of the ladder, which runs between the upper and lower hatches.

- Use CFD to generate fire scenarios, which will aid in determining the manning requirements for DD-21, which is still in the design phase

- Develop a scenario which has a fire within the modeled space to determine its effects. Add on adjacent spaces to see how the fire will affect them.

- Use CFD to compute the time it takes the wall to reach a given temperature using the transient function that will be available in version 6.0

THIS PAGE INTENTIONALLY LEFT BLANK

APPENDIX A.

Scenario A was developed as a control Scenario. The results of this scenario will be compared to Scenarios B and C. The inlet is located at the X-max face with the upper half of the door designated as the smoke inlet, while the lower half is exclusively air. Refer to the next page for the required inputs.

Relaxation	Velocity	(m/s)	0.2
	Turbulence	(J)	1.0
	Enthalpy	(KJ/kg)	1.0
	Mixture Fractions		0.2
Sweeps	Velocity	(m/s)	5
	Pressure	(Pa)	30
Initial Conditions	U Velocity	(m/s)	-0.1
	V Velocity	(m/s)	0
	W Velocity	(m/s)	0
	Relative pressure	(Pa)	0
	Turbulence Kinetic Energy	(J)	0.04
	Rate of Turbulence Dissipation	(J/s)	-0.05
	Turbulent Length Scale	(m)	0.06
	Temperature	(K)	500
	Reference Pressure	(Pa)	1E5
Boundary Conditions	Isothermal Wall Temperature	(K)	300
Inlet – Smoke	U Velocity	(m/s)	-0.1
	V Velocity	(m/s)	0
	W Velocity	(m/s)	0
	Temperature	(K)	500
	Turbulence Kinetic Energy	(J)	0.
	Rate of Turbulence Dissipation	(J/s)	0.04
	Turbulence Length Scale	(m)	0.06
	Pressure	(Pa)	0
Inlet - Air	U Velocity	(m/s)	-0.1
	V Velocity	(m/s)	0
	W Velocity	(m/s)	0
	Temperature	(K)	500
	Turbulence Kinetic Energy	(J)	0.04
	Rate of Turbulence Dissipation	(J/s)	-0.05
	Turbulence Length Scale	(m)	0.06
	Pressure	(Pa)	0

Table 1. Input Data for Scenario A (See Chapter III for definitions)

Outlet	U Velocity	(m/s)	0
	V Velocity	(m/s)	0
	W Velocity	(m/s)	0
	Temperature	(K)	400
	Turbulence Kinetic Energy	(J)	0.02
	Rate of Turbulence Dissipation	(J/s)	-0.05
	Turbulence Length Scale	(m)	0.08
	Pressure	(Pa)	0

Table 1 Cont. Input Data for Scenario A (See Chapter III for definitions)
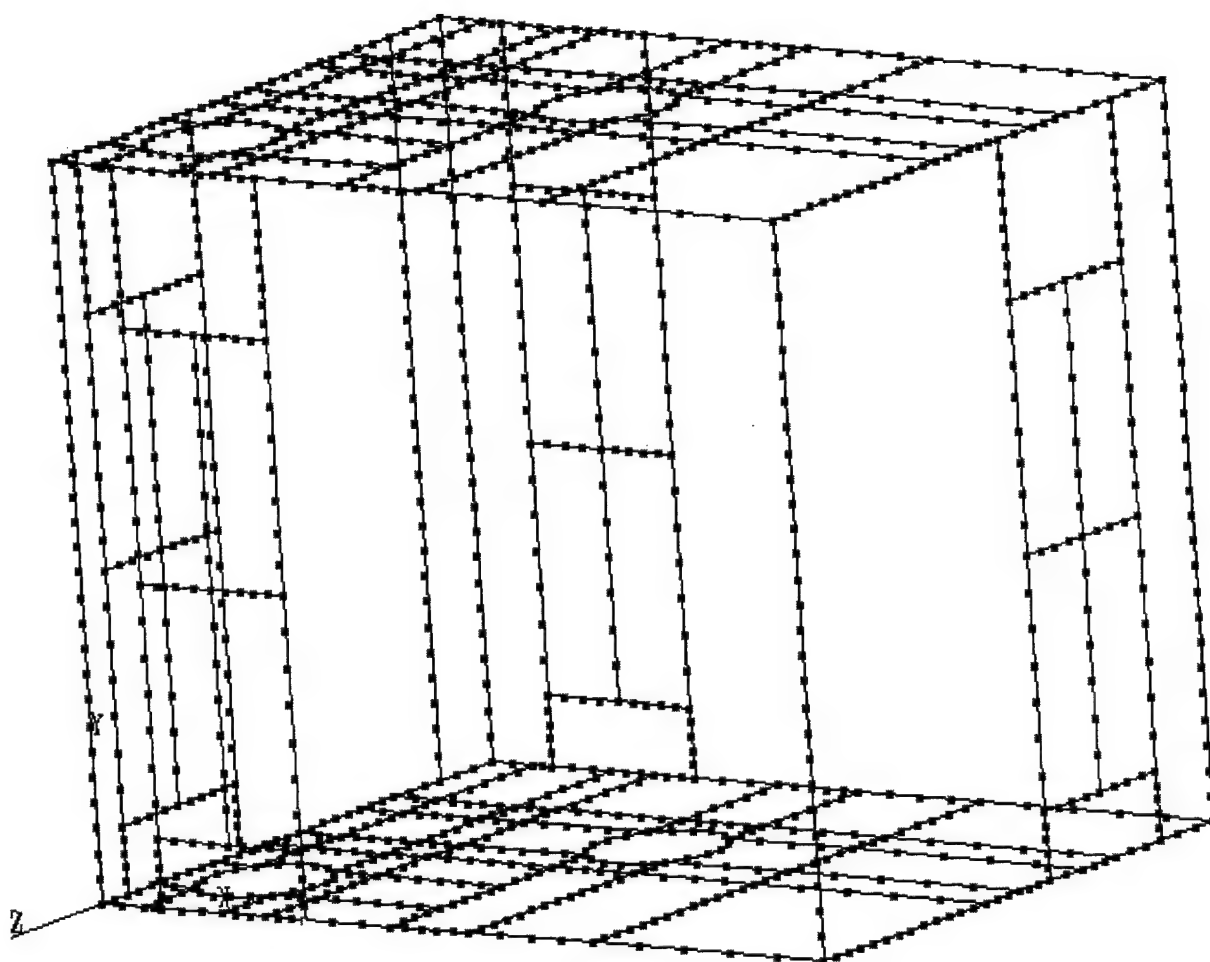
Figure 9. Three dimensional design of Inlet (Red door on Right) and Outlet (Red Door on Left). Top Half of Inlet is Smoke while Bottom Half is Air.
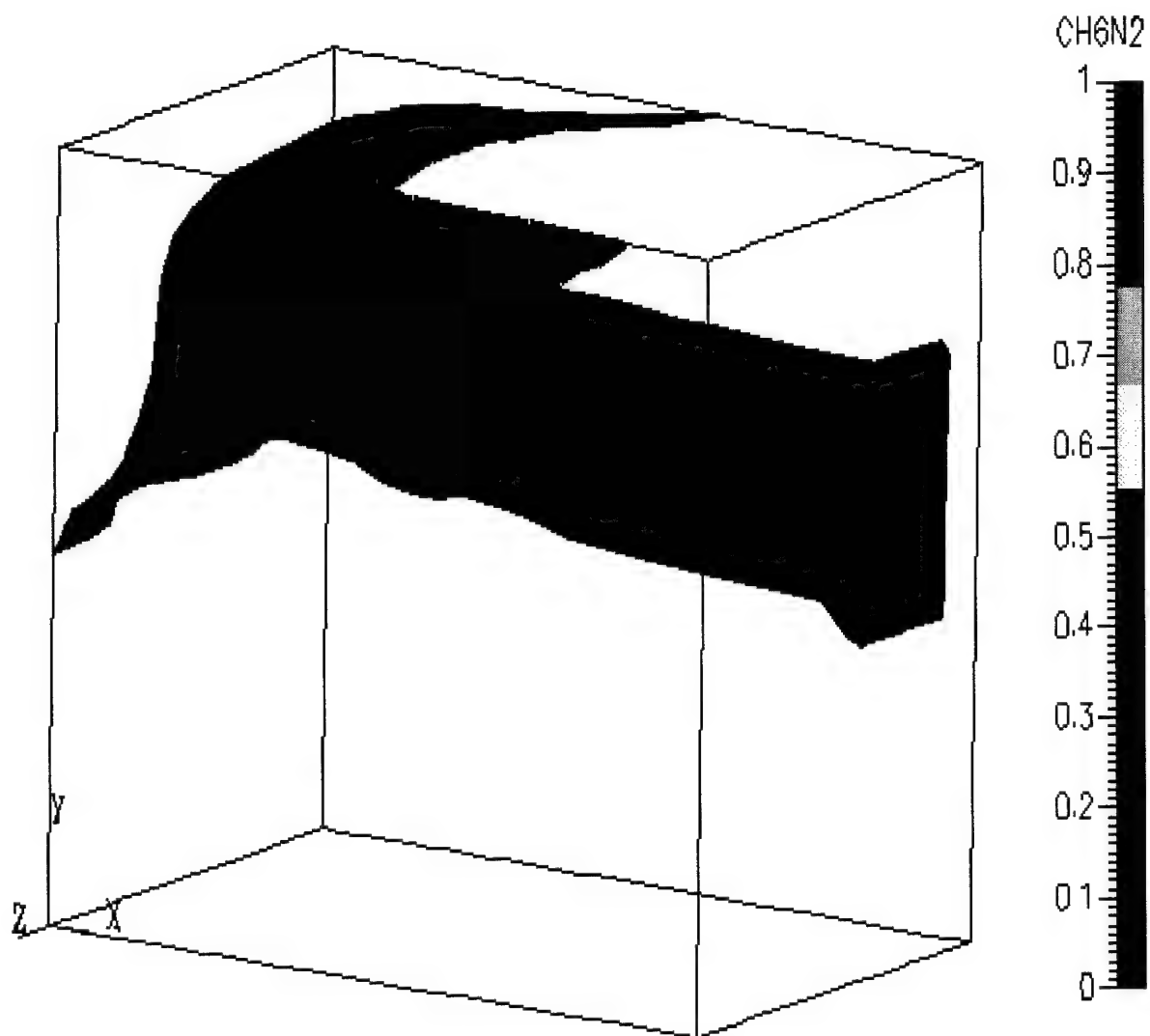
Figure 10. Isosurface of Smoke showing 98 percent concentration by volume
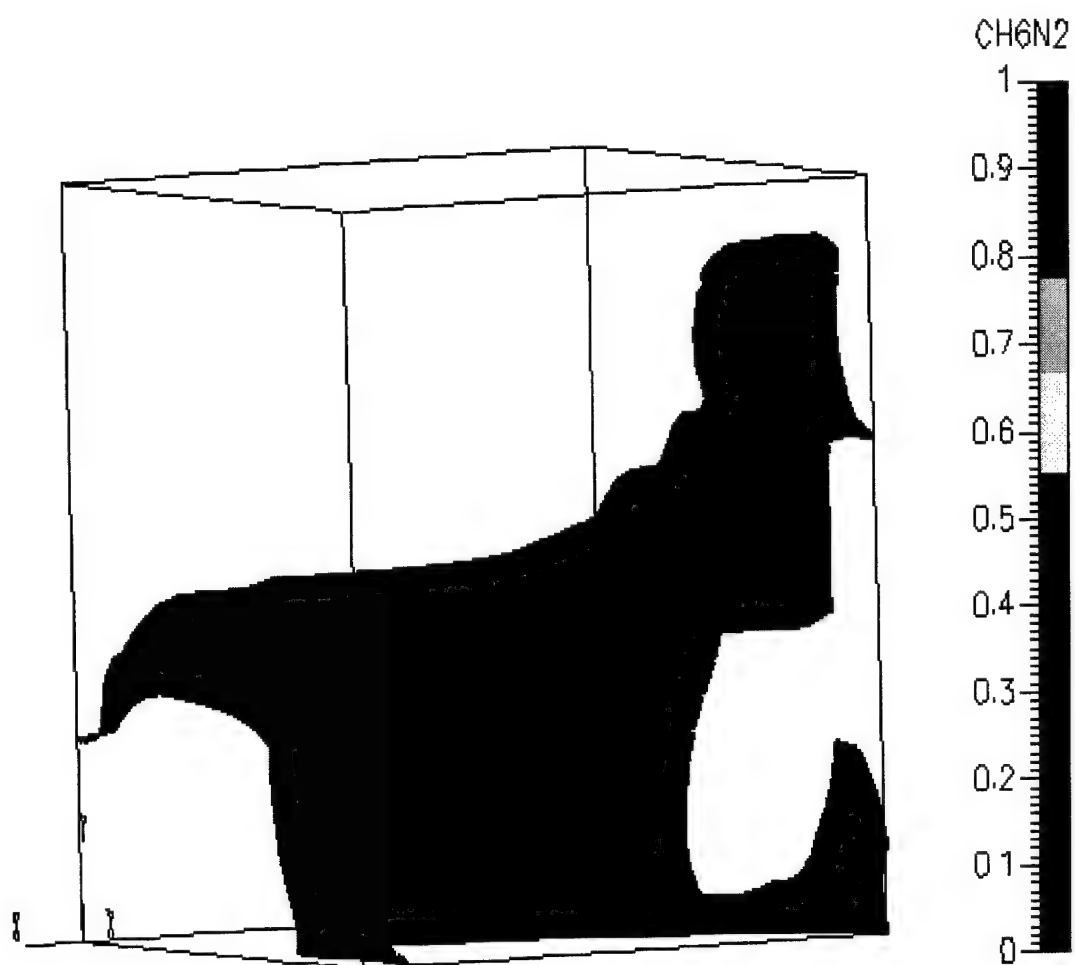
Figure 11. Isosurface of Smoke showing 54 percent concentration by volume
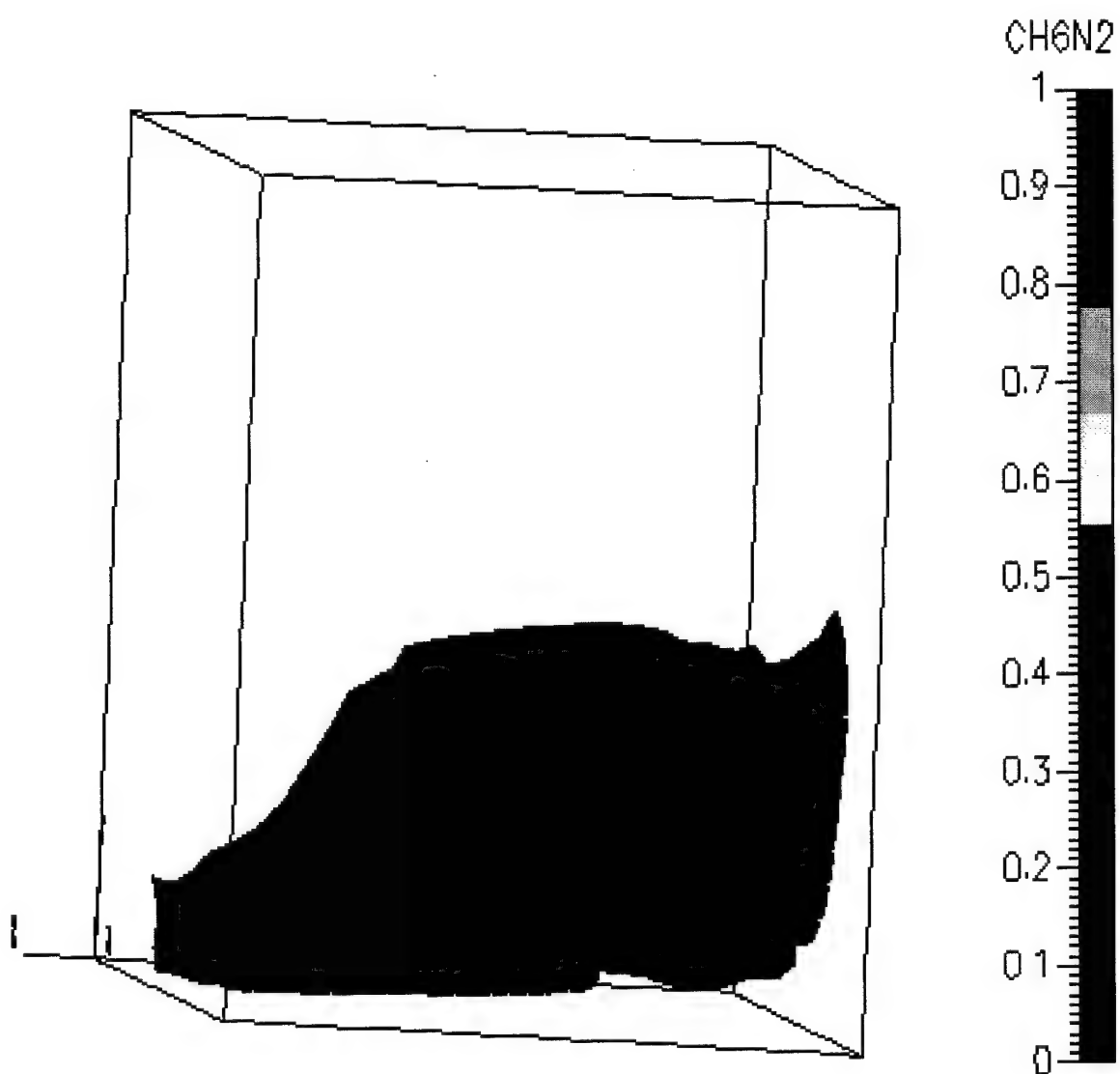
Figure 12. Isosurface of Smoke showing 22 percent by volume
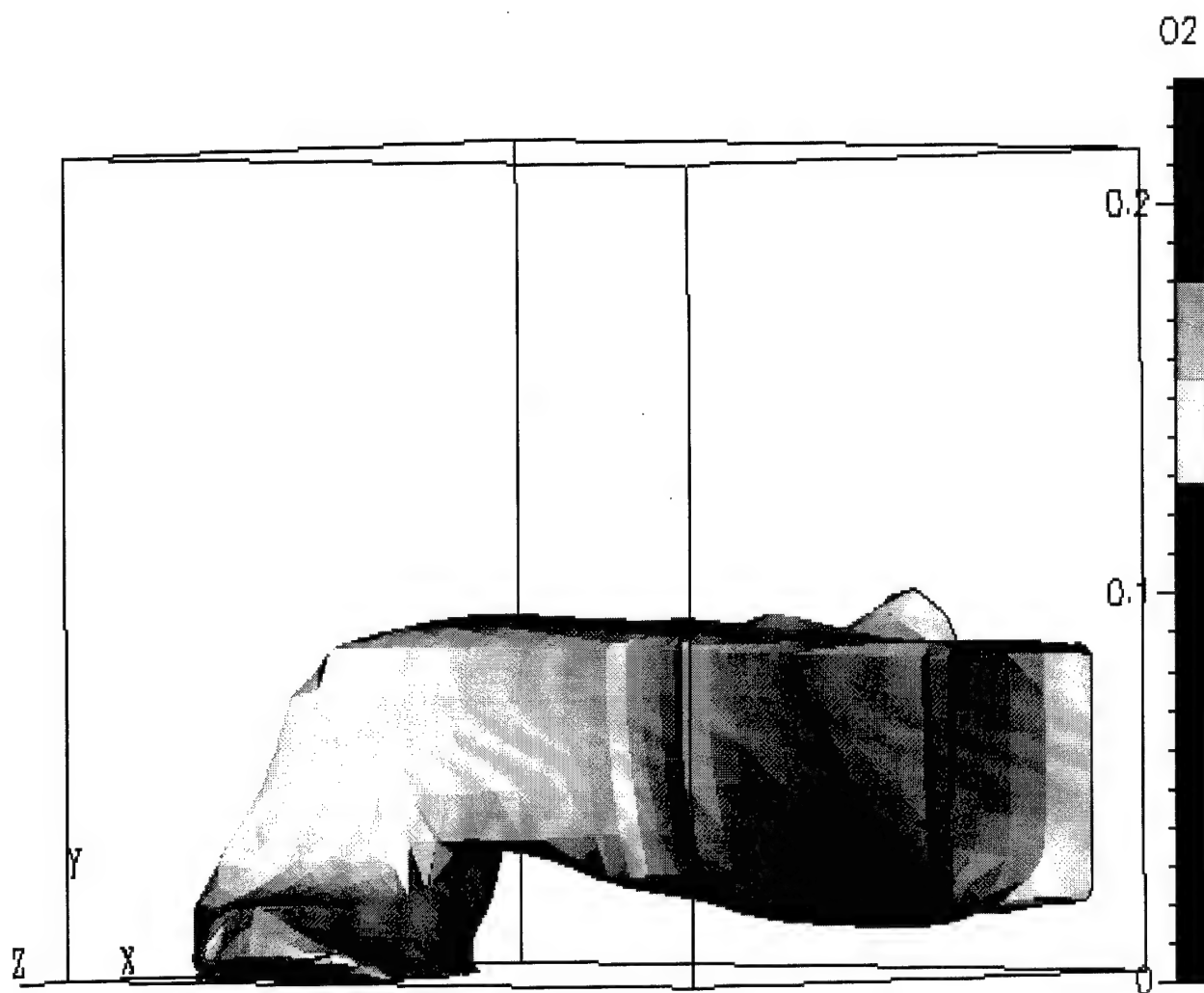
Figure 13. Isosurface of Air showing 77 percent by volume
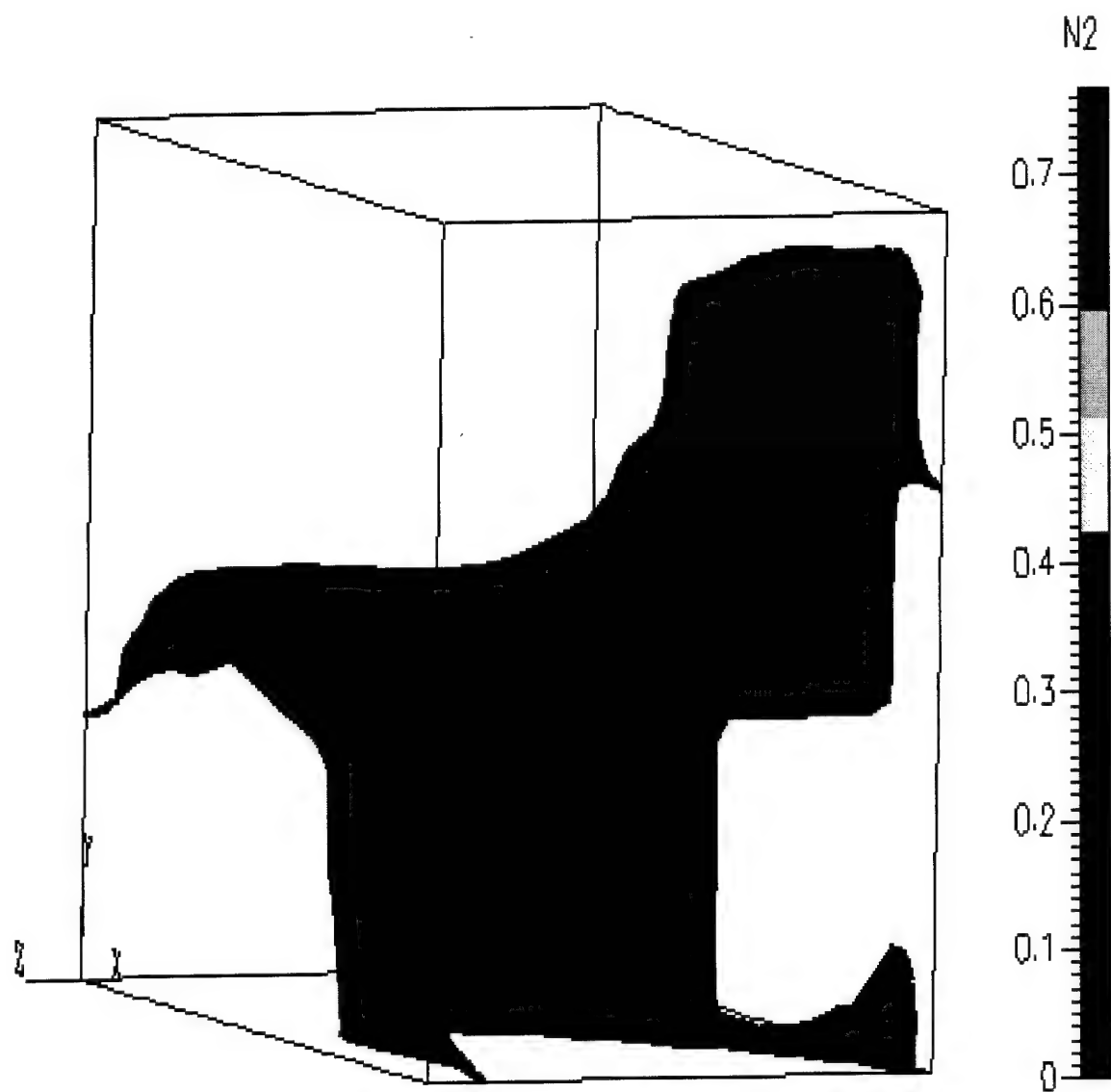
Figure 14. Isosurface of Air at 54 percent concentration by volume
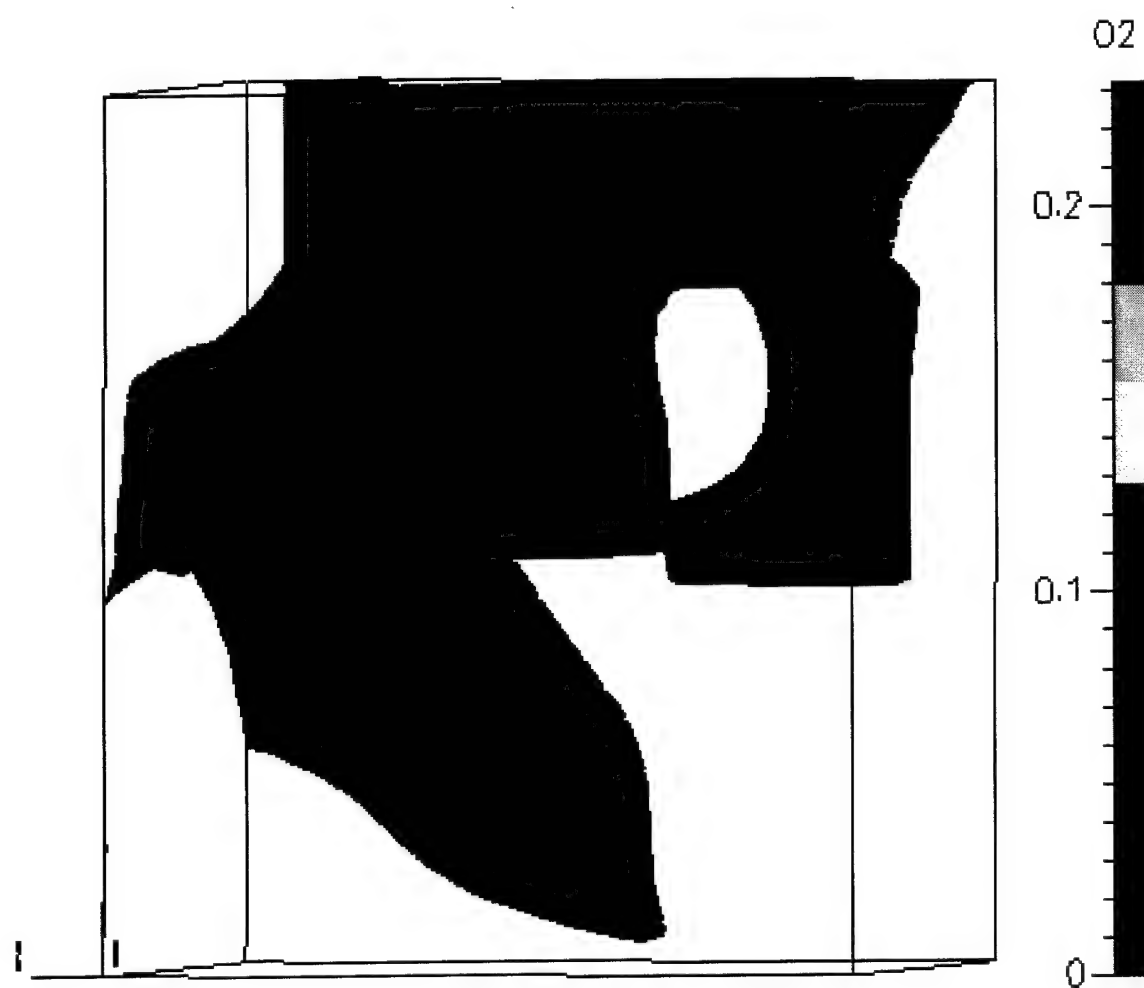
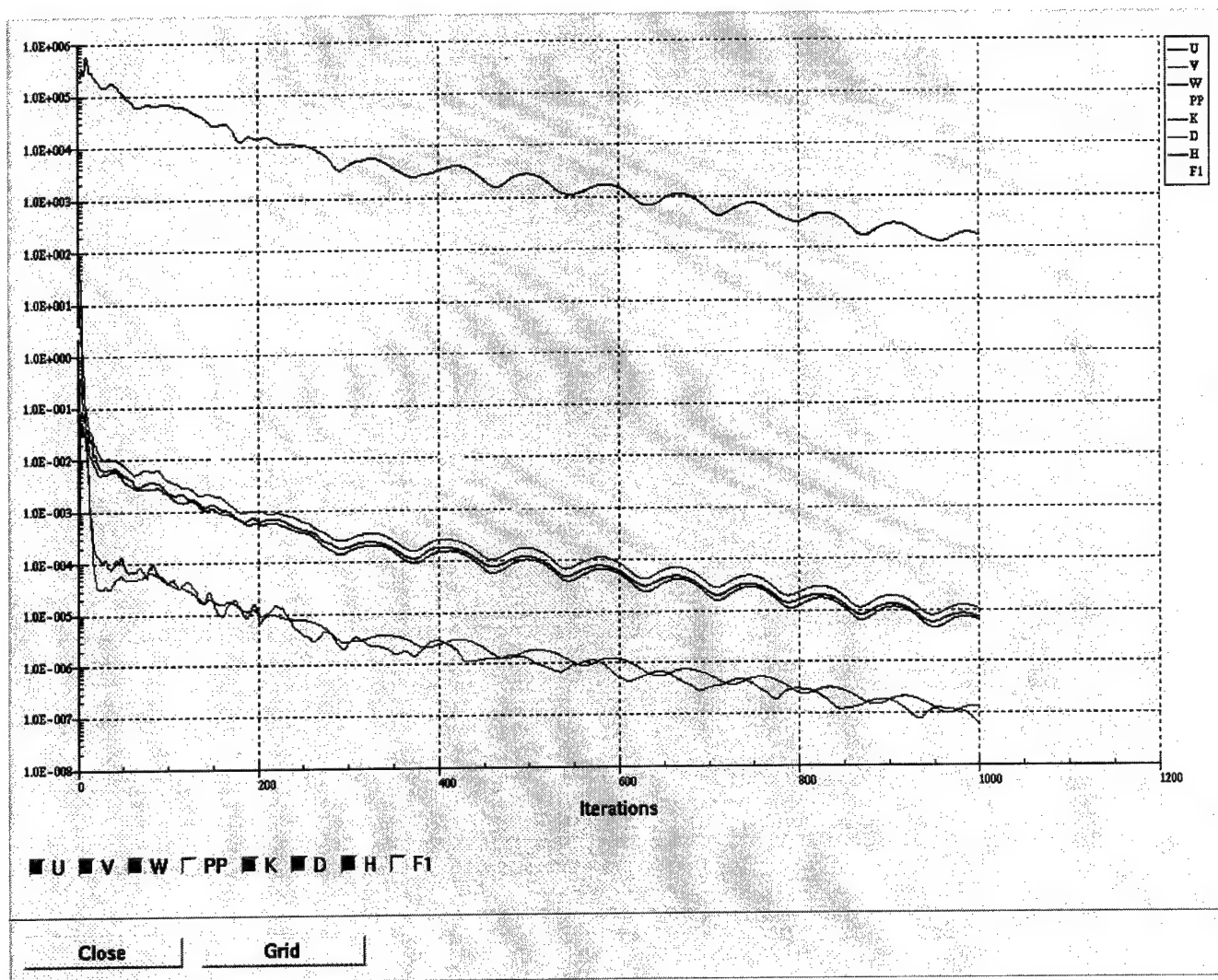**Figure 15. Isosurface of Air at 16 percent concentration by volume**

Figure 16. Residuals for Scenario A

THIS PAGE INTENTIONALLY LEFT BLANK

APPENDIX B

Scenario B was developed to contrast with Scenario A. The inlet is located at the X-max face with the upper half of the door designated as the smoke inlet, while the lower half is exclusively air. The major difference between this model and the others is that the smoke is 500 K, while the air is 300 K. Refer to the next page for the required inputs.

Relaxation	Velocity	(m/s)	0.2
	Turbulence	(J)	1.0
	Enthalpy	(KJ/kg)	1.0
	Mixture Fractions		0.2
Sweeps	Velocity	(m/s)	5
	Pressure	(Pa)	30
Initial Conditions	U Velocity	(m/s)	-0.1
	V Velocity	(m/s)	0
	W Velocity	(m/s)	0
	Relative pressure	(Pa)	0
	Turbulence Kinetic Energy	(J)	0.04
	Rate of Turbulence Dissipation	(J/s)	-0.05
	Turbulent Length Scale	(m)	0.06
	Temperature	(K)	400
	Reference Pressure	(Pa)	1E5
Boundary Conditions	Isothermal Wall Temperature	(K)	300
Inlet – Smoke	U Velocity	(m/s)	-0.1
	V Velocity	(m/s)	0
	W Velocity	(m/s)	0
	Temperature	(K)	500
	Turbulence Kinetic Energy	(J)	0.04
	Rate of Turbulence Dissipation	(J/s)	-0.05
	Turbulence Length Scale	(m)	0.06
	Pressure	(Pa)	0
Inlet - Air	U Velocity	(m/s)	-0.1
	V Velocity	(m/s)	0
	W Velocity	(m/s)	0
	Temperature	(K)	300
	Turbulence Kinetic Energy	(J)	0.04
	Rate of Turbulence Dissipation	(J/s)	-0.05
	Turbulence Length Scale	(m)	0.06
	Pressure	(Pa)	0

Table 2. Input Data for Scenario B (See Chapter III for definitions)

Outlet	U Velocity	(m/s)	0
	V Velocity	(m/s)	0
	W Velocity	(m/s)	0
	Temperature	(K)	400
	Turbulence Kinetic Energy	(J)	0.02
	Rate of Turbulence Dissipation	(J/s)	-0.05
	Turbulence Length Scale	(m)	0.08
	Pressure	(Pa)	0

Table 2 Cont. Input Data for Scenario B (See Chapter III for definitions)
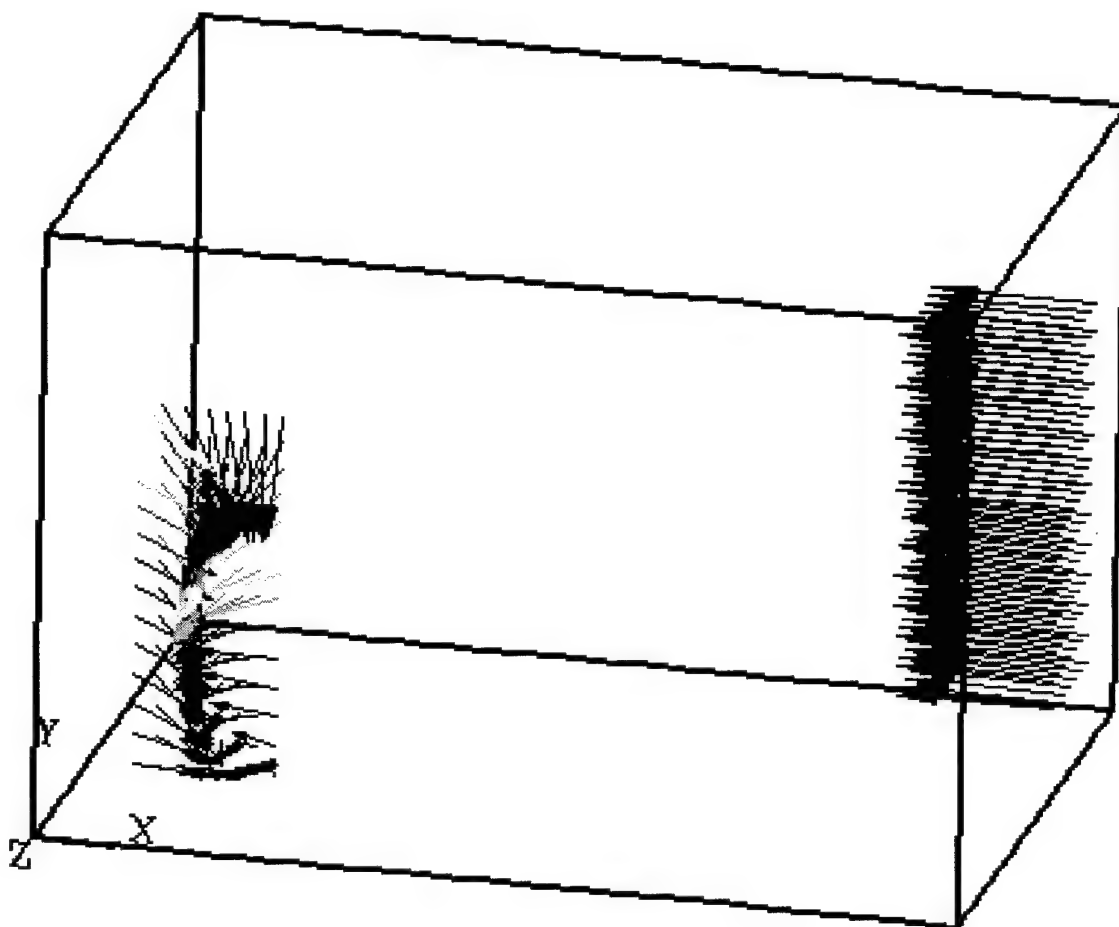
Figure 17. The Inlet is on the right with smoke (blue) as the upper fluid at 500 K and air (red) on the bottom at 300 K. The Outlet is shown on the left.
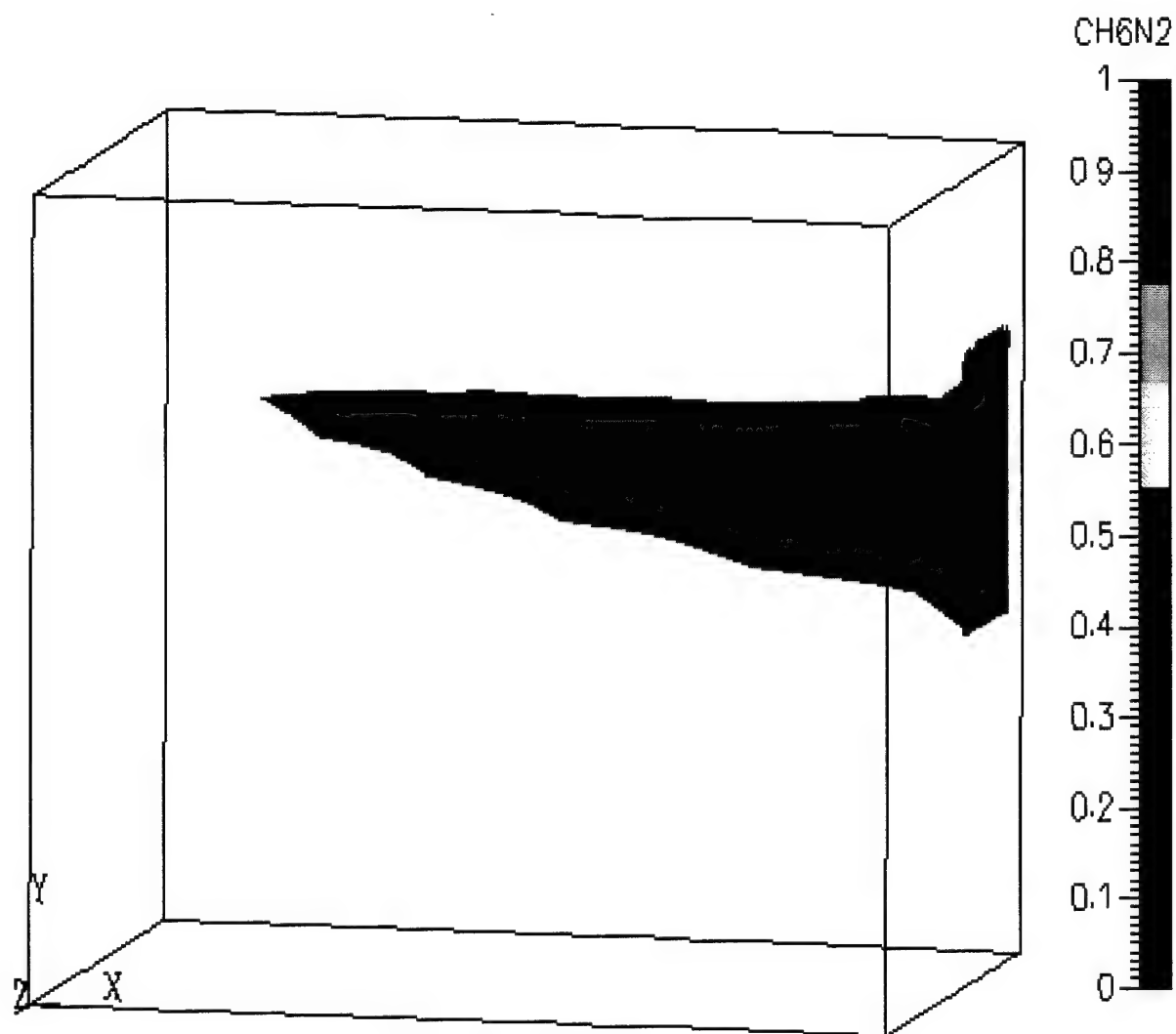
Figure 18. Isosurface of Smoke showing 98 percent concentration by volume
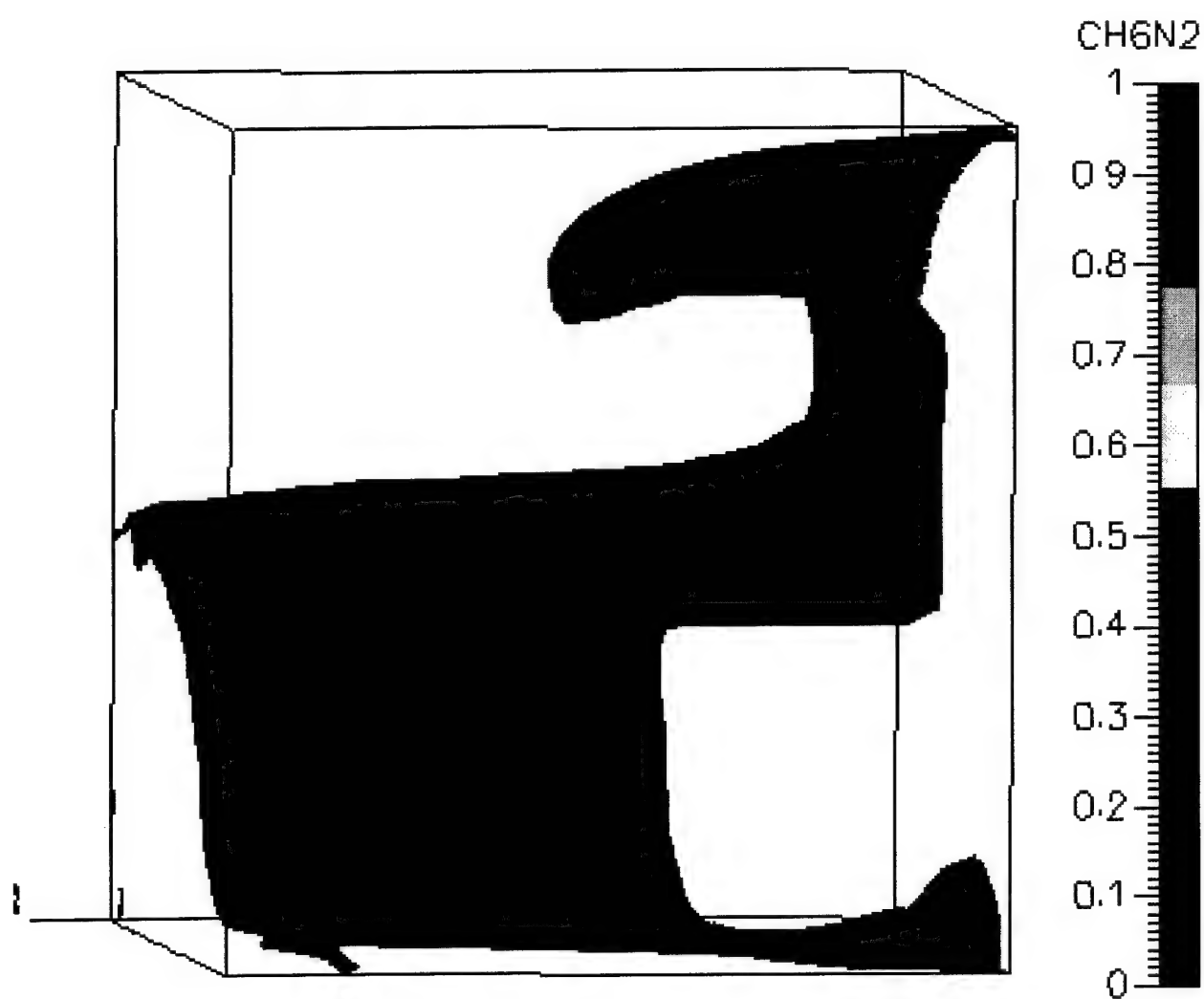
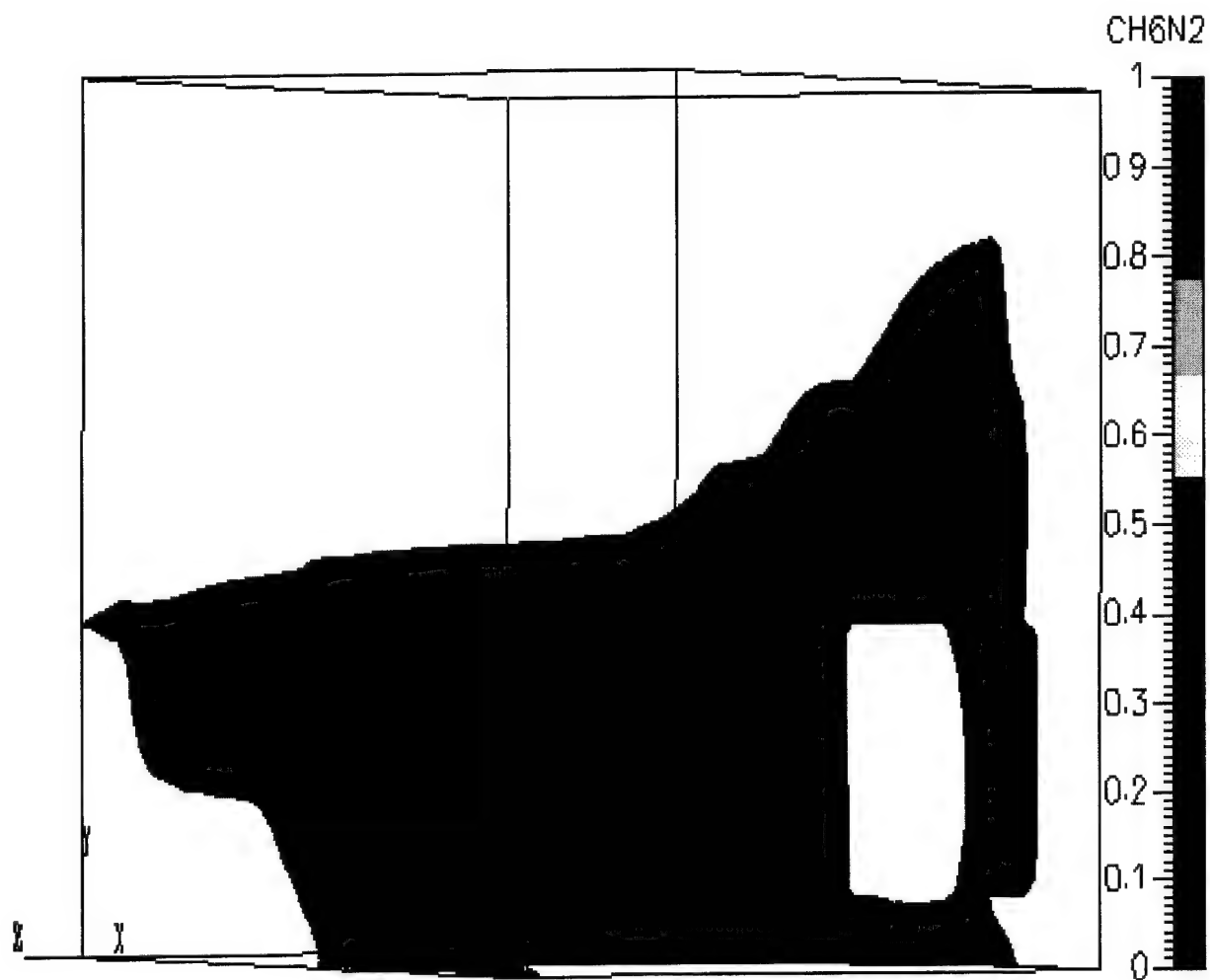**Figure 19. Isosurface of Smoke showing 54 percent concentration by volume**

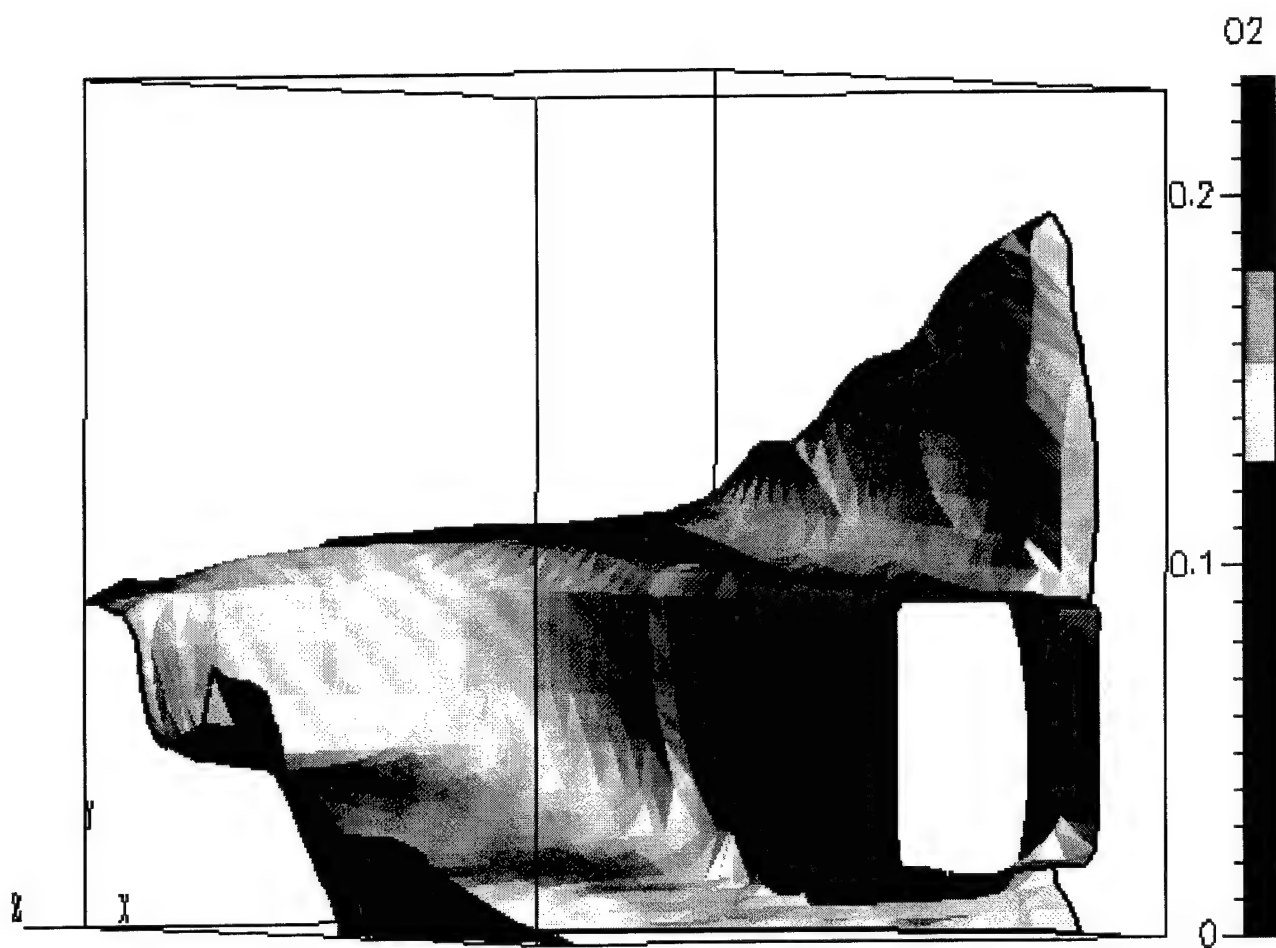**Figure 20. Isosurface of Smoke showing 22 percent concentration by volume**

Figure 21. Isosurface of Air at 77 percent concentration by volume

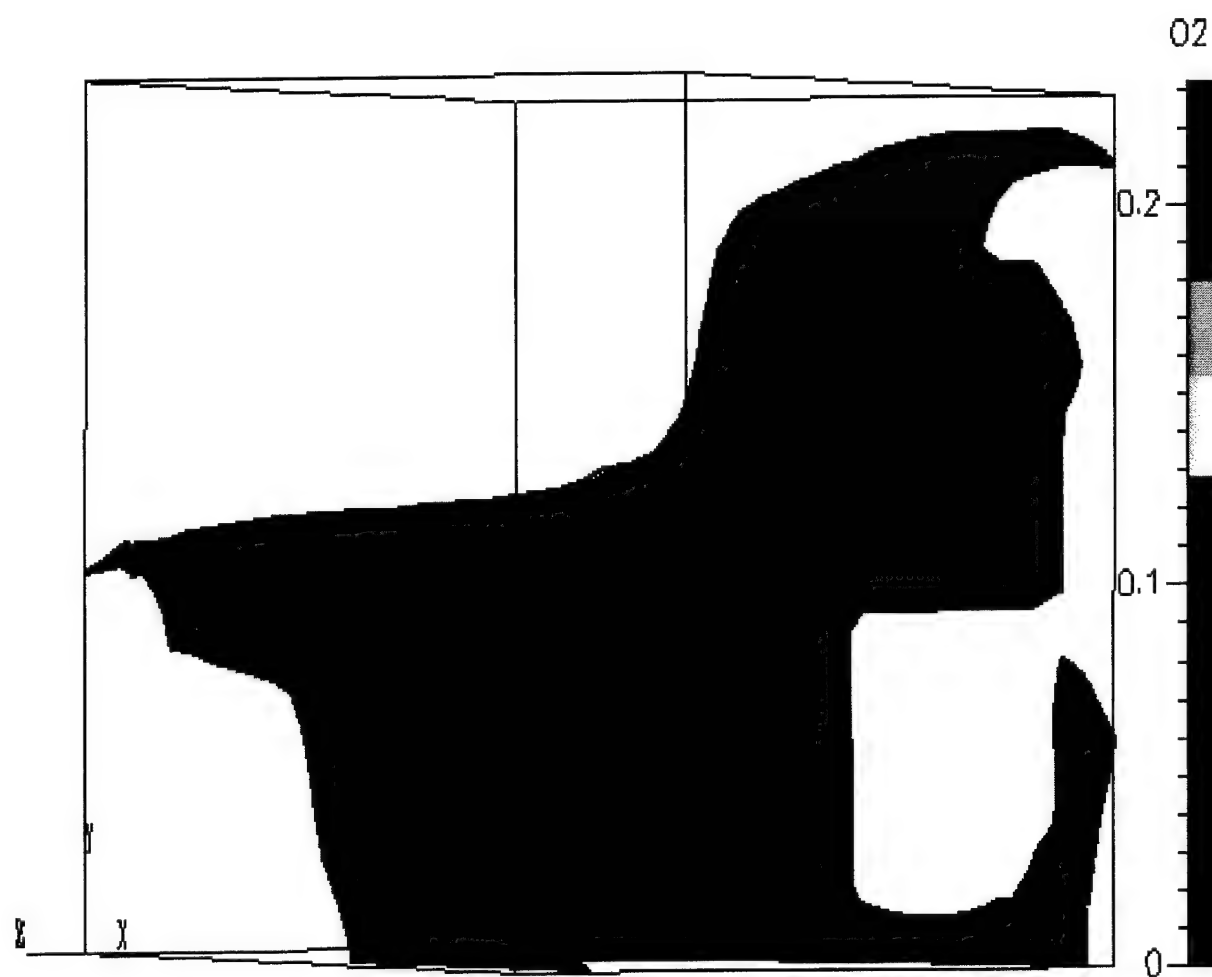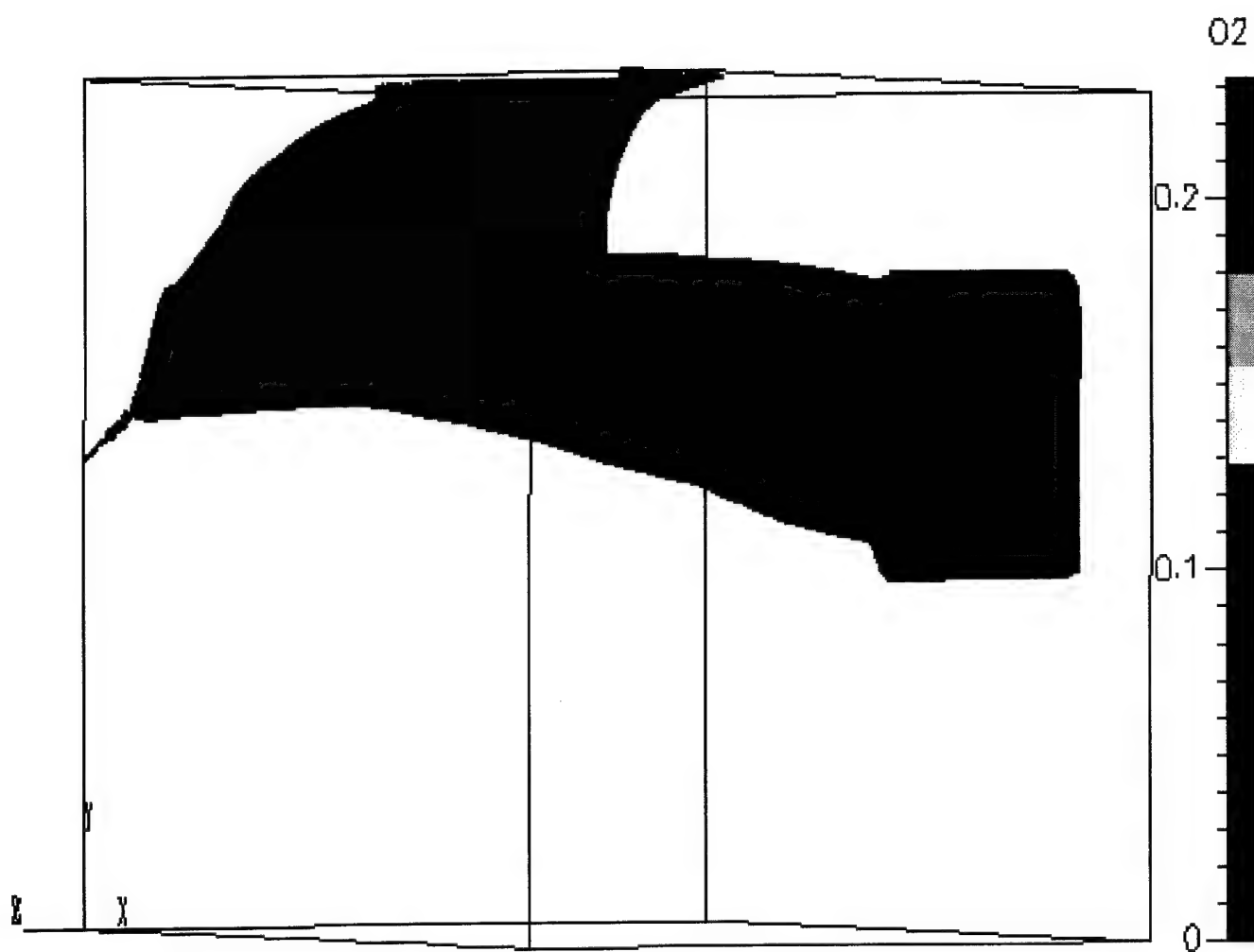**Figure 22. Isosurface of Air at 54 percent concentration by volume**

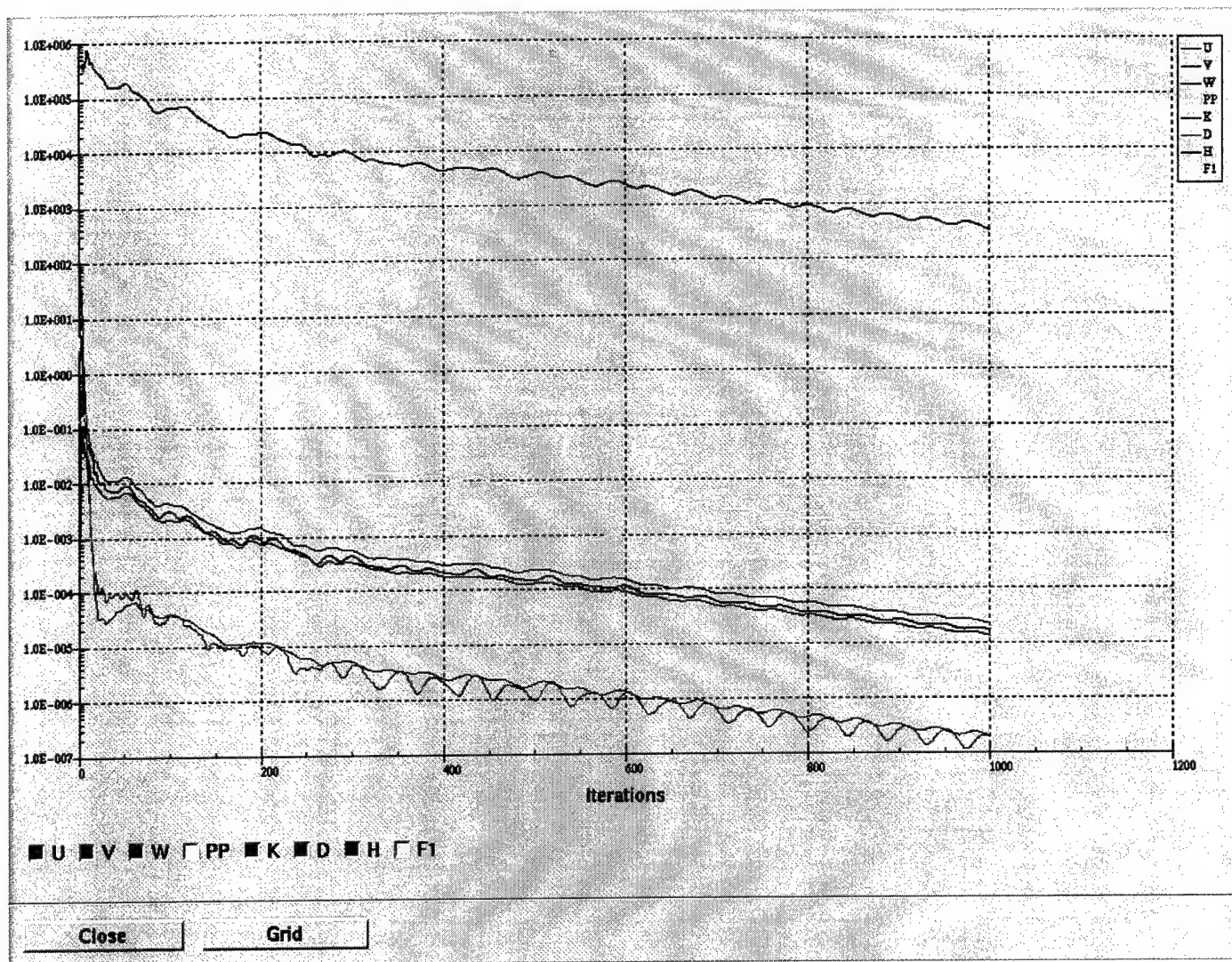**Figure 23. Isosurface of Air at 16 percent concentration by volume**

Figure 24. Residuals for Scenario B

THIS PAGE INTENTIONALLY LEFT BLANK

APPENDIX C

Scenario C was developed to contrast with Scenarios A and B. The inlet is located at the X-max face with the upper half of the door designated as the smoke inlet, while the lower half is exclusively air. Both fluids enter at 500 K. The Y-min face is held isothermally at 1200 K. Refer to the next page for the required inputs.

Relaxation	Velocity	(m/s)	0.2
	Turbulence	(J)	1.0
	Enthalpy	(KJ/kg)	1.0
	Mixture Fractions		0.2
Sweeps	Velocity	(m/s)	5
	Pressure	(Pa)	30
Initial Conditions	U Velocity	(m/s)	-0.1
	V Velocity	(m/s)	0
	W Velocity	(m/s)	0
	Relative pressure	(Pa)	0
	Turbulence Kinetic Energy	(J)	0.04
	Rate of Turbulence Dissipation	(J/s)	-0.05
	Turbulent Length Scale	(m)	0.06
	Temperature	(K)	500
	Reference Pressure	(Pa)	1E5
Boundary Conditions	Isothermal Floor Temperature	(K)	1200
	Isothermal Wall Temperature	(K)	300
Inlet – Smoke	U Velocity	(m/s)	-0.1
	V Velocity	(m/s)	0
	W Velocity	(m/s)	0
	Temperature	(K)	500
	Turbulence Kinetic Energy	(J)	0.04
	Rate of Turbulence Dissipation	(J/s)	-0.05
	Turbulence Length Scale	(m)	0.06
	Pressure	(Pa)	0
Inlet - Air	U Velocity	(m/s)	-0.1
	V Velocity	(m/s)	0
	W Velocity	(m/s)	0
	Temperature	(K)	500
	Turbulence Kinetic Energy	(J)	0.04
	Rate of Turbulence Dissipation	(J/s)	-0.05
	Turbulence Length Scale	(m)	0.06
	Pressure	(Pa)	0

Table 3. Input Data for Scenario C (See Chapter III for definitions)

Outlet	U Velocity	(m/s)	0
	V Velocity	(m/s)	0
	W Velocity	(m/s)	0
	Temperature	(K)	400
	Turbulence Kinetic Energy	(J)	0.02
	Rate of Turbulence Dissipation	(J/s)	-0.05
	Turbulence Length Scale	(m)	0.08
	Pressure	(Pa)	0

Table 3 Cont. Input Data for Scenario C (See Chapter III for definitions)
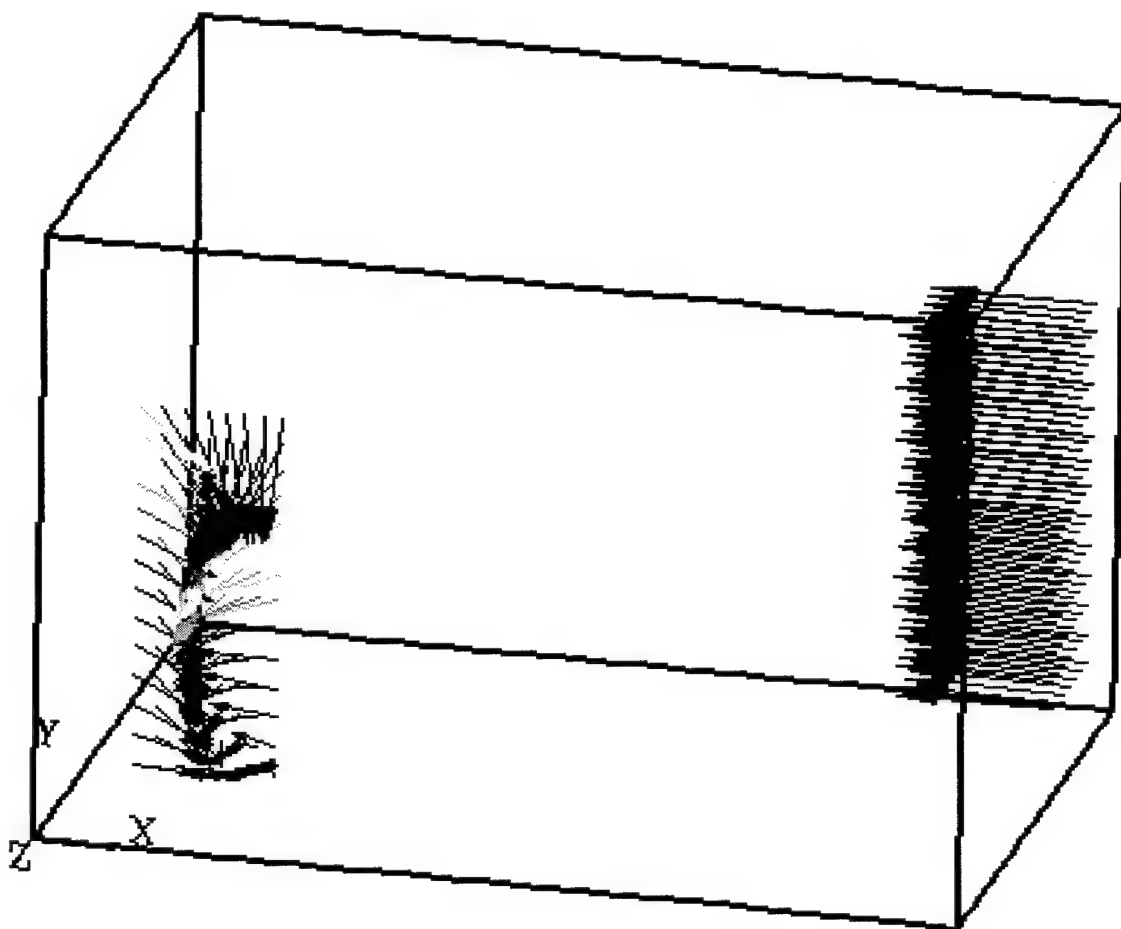
Figure 25. The Inlet is on the right with smoke (blue) as the upper fluid at 500 K and air (red) on the bottom at 500 K. The Outlet is shown on the left.
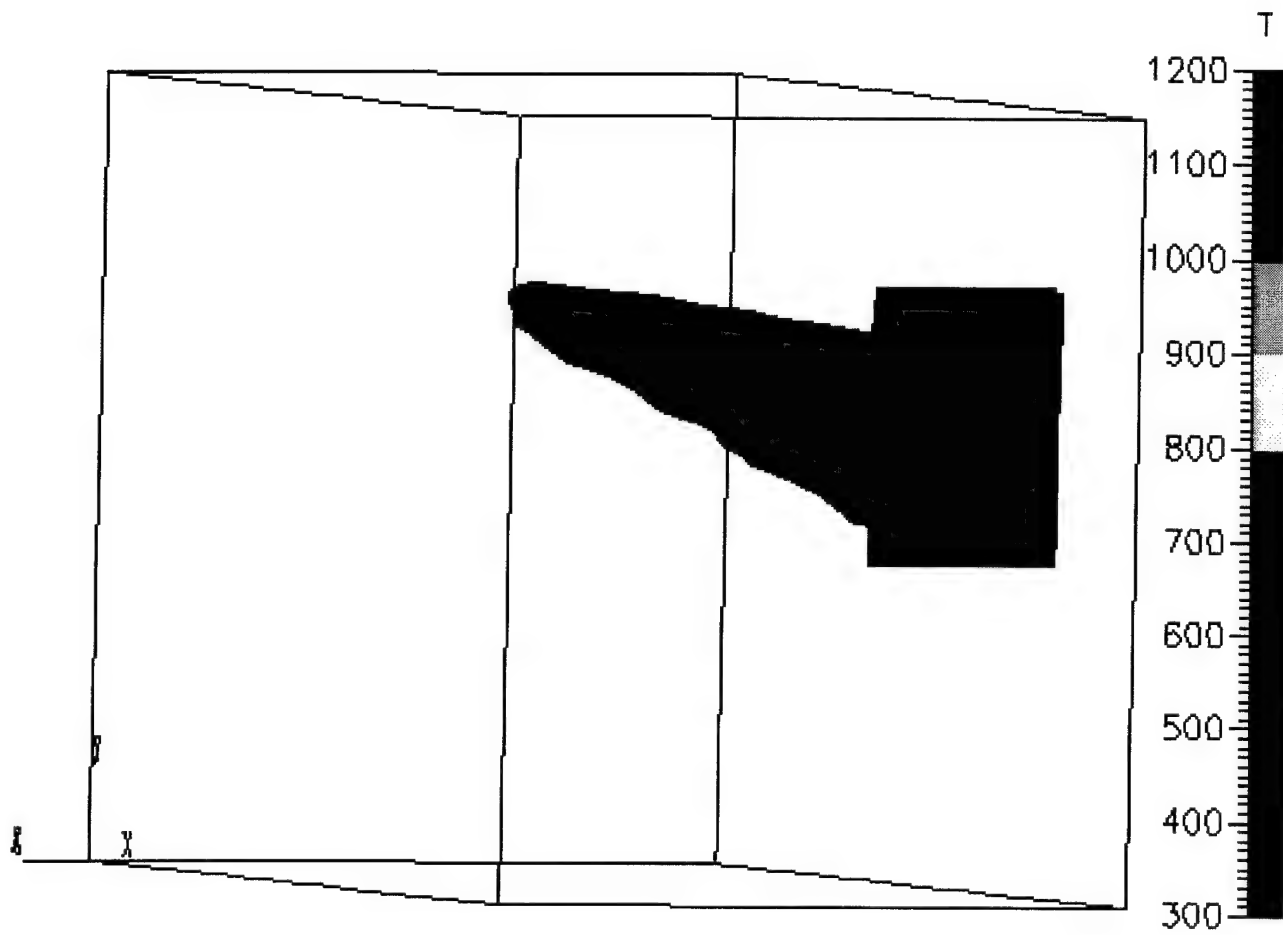
Figure 26. Isosurface of Smoke showing 98 percent concentration by volume.
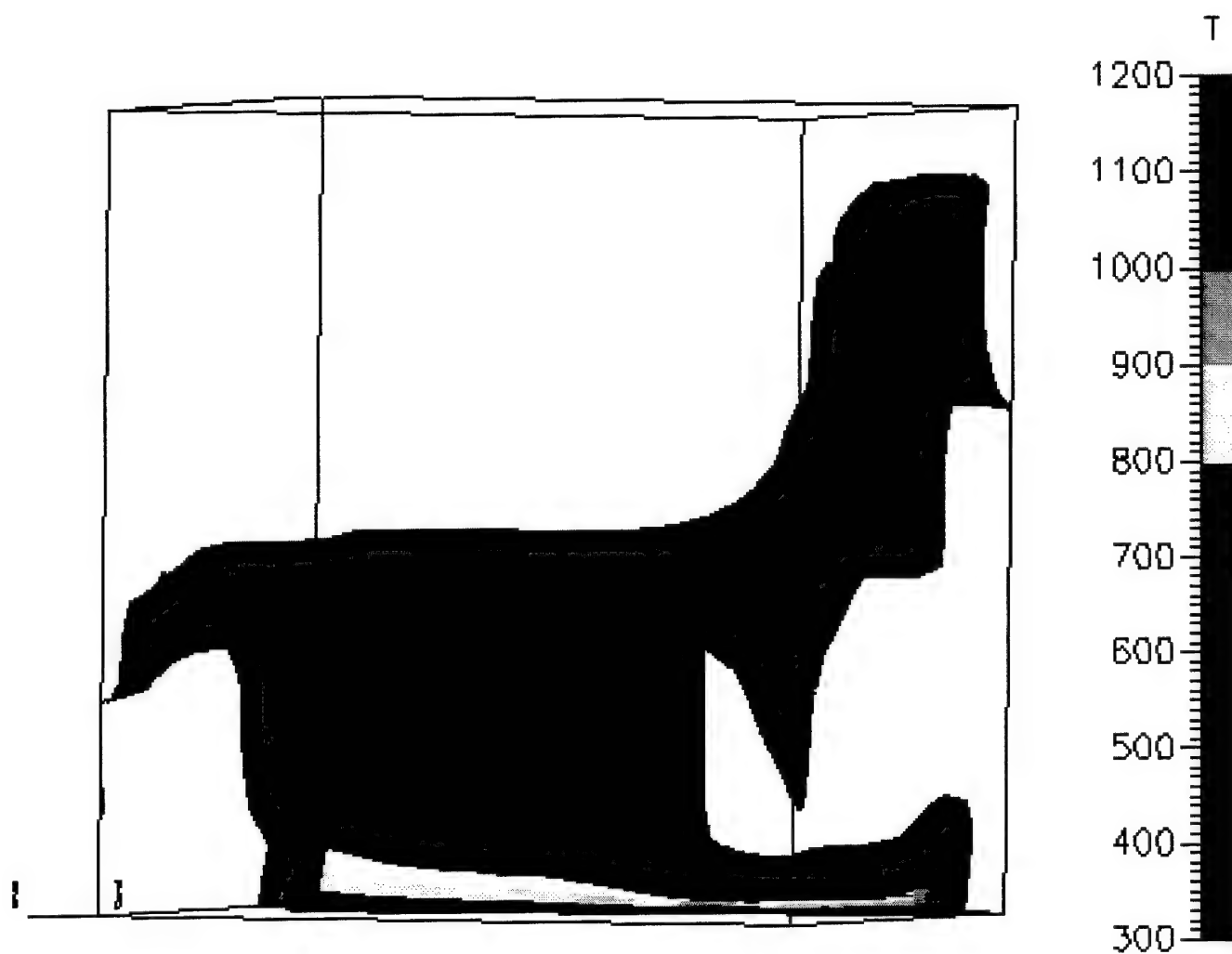
Figure 27. Isosurface of Smoke showing 54 percent concentration by volume.
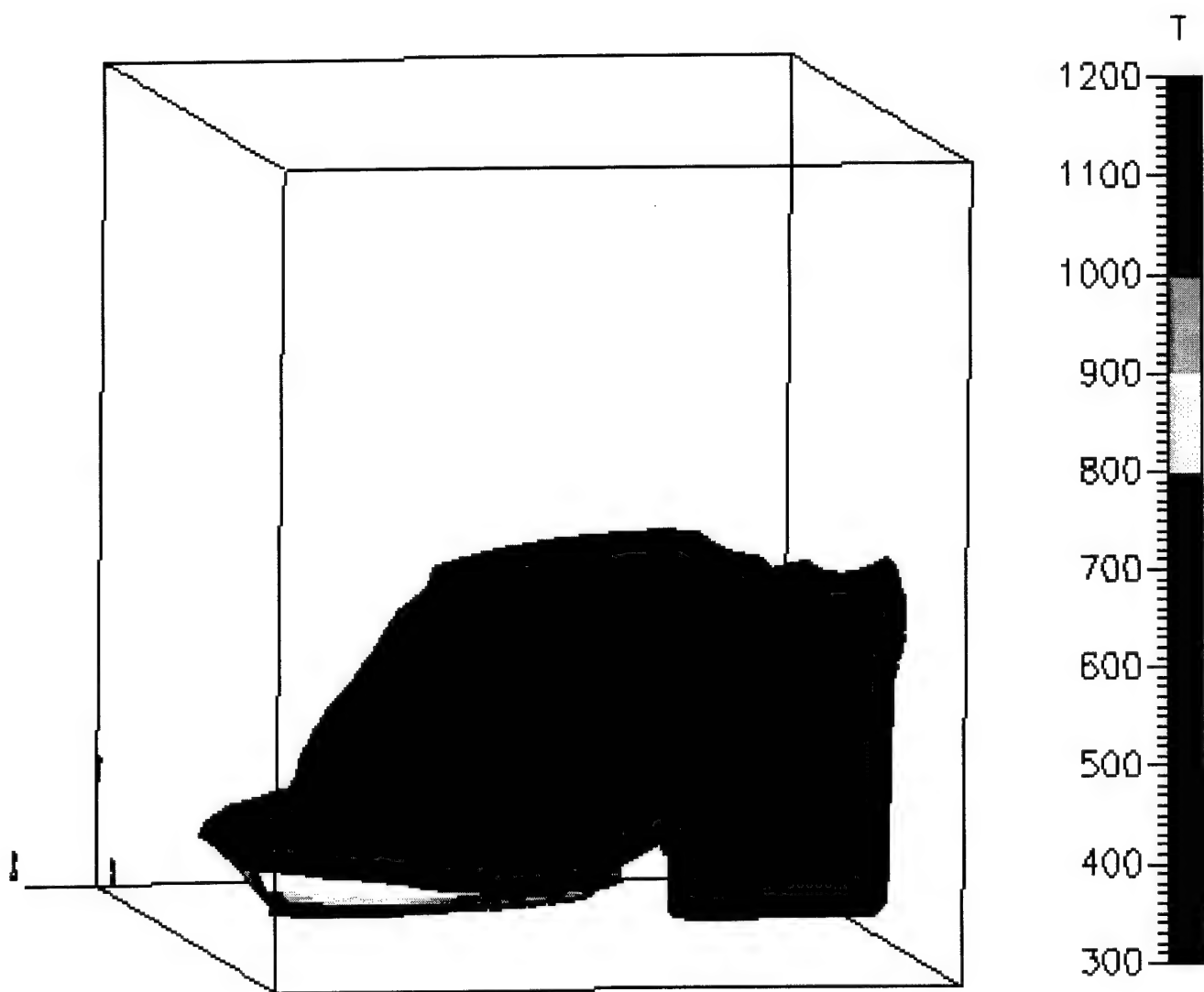
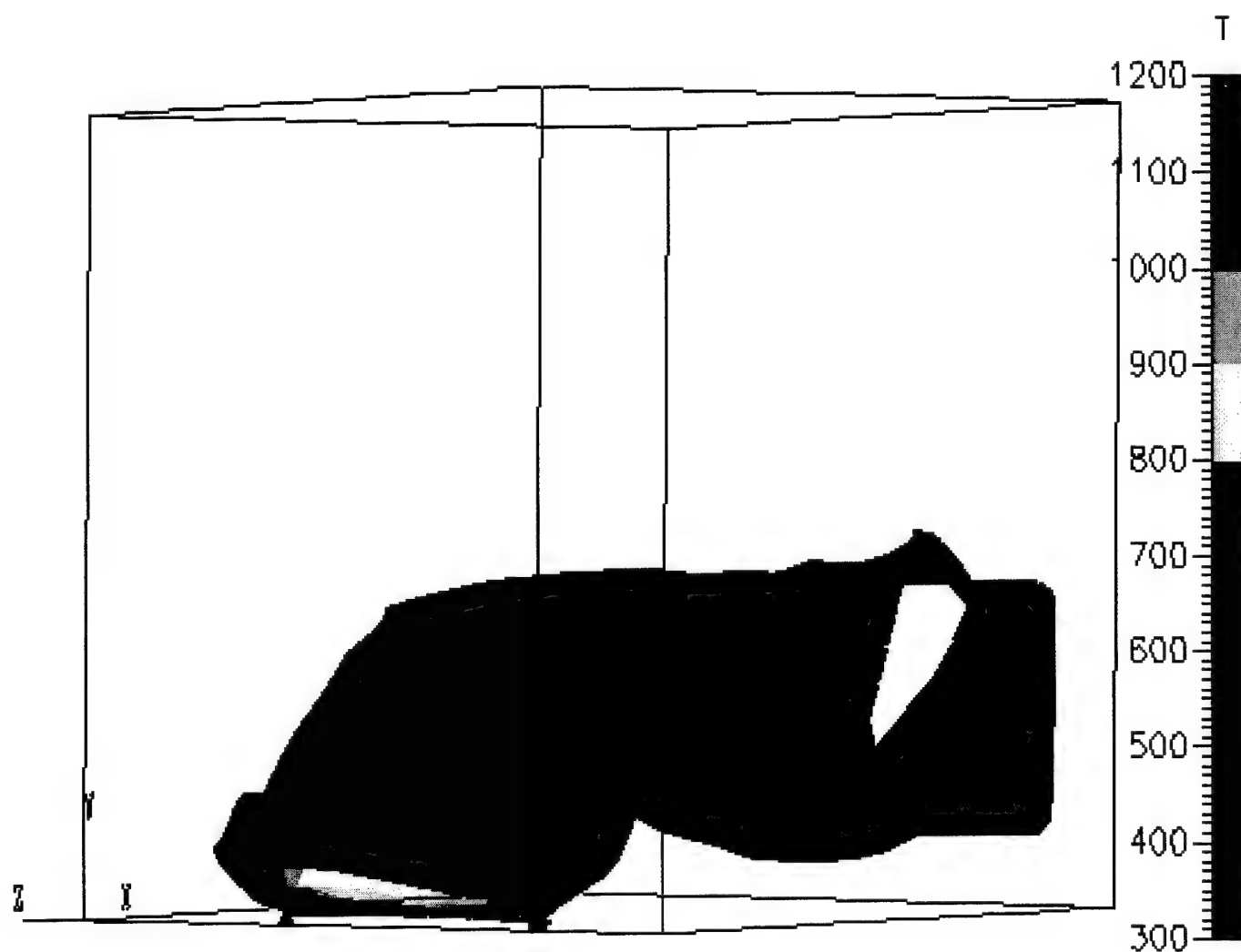**Figure 28. Isosurface of Smoke showing 22 percent concentration by volume.**

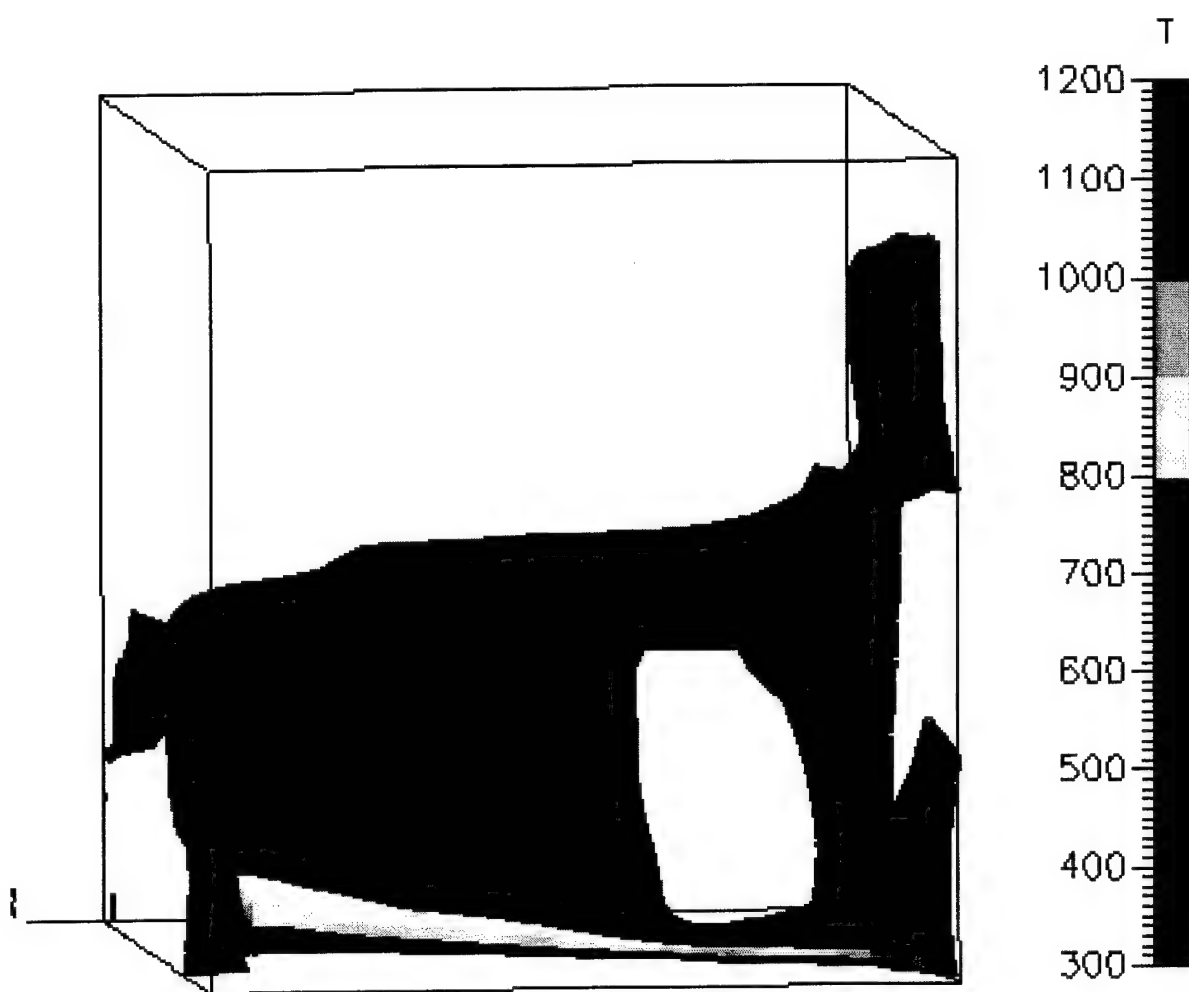**Figure 29. Isosurface of Air showing 77 percent concentration by volume.**

68

Figure 30. Isosurface of Smoke showing 54 percent concentration by volume.
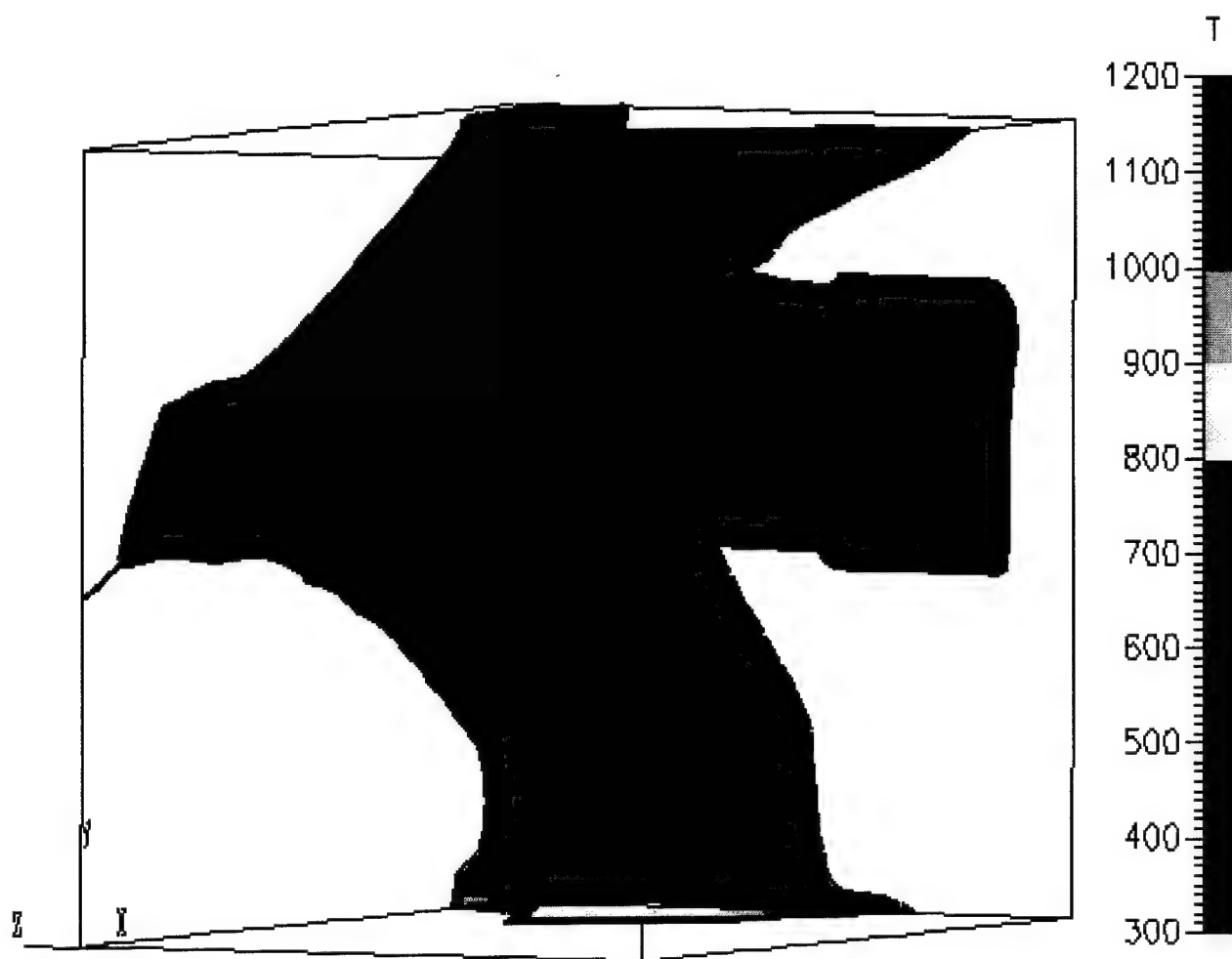
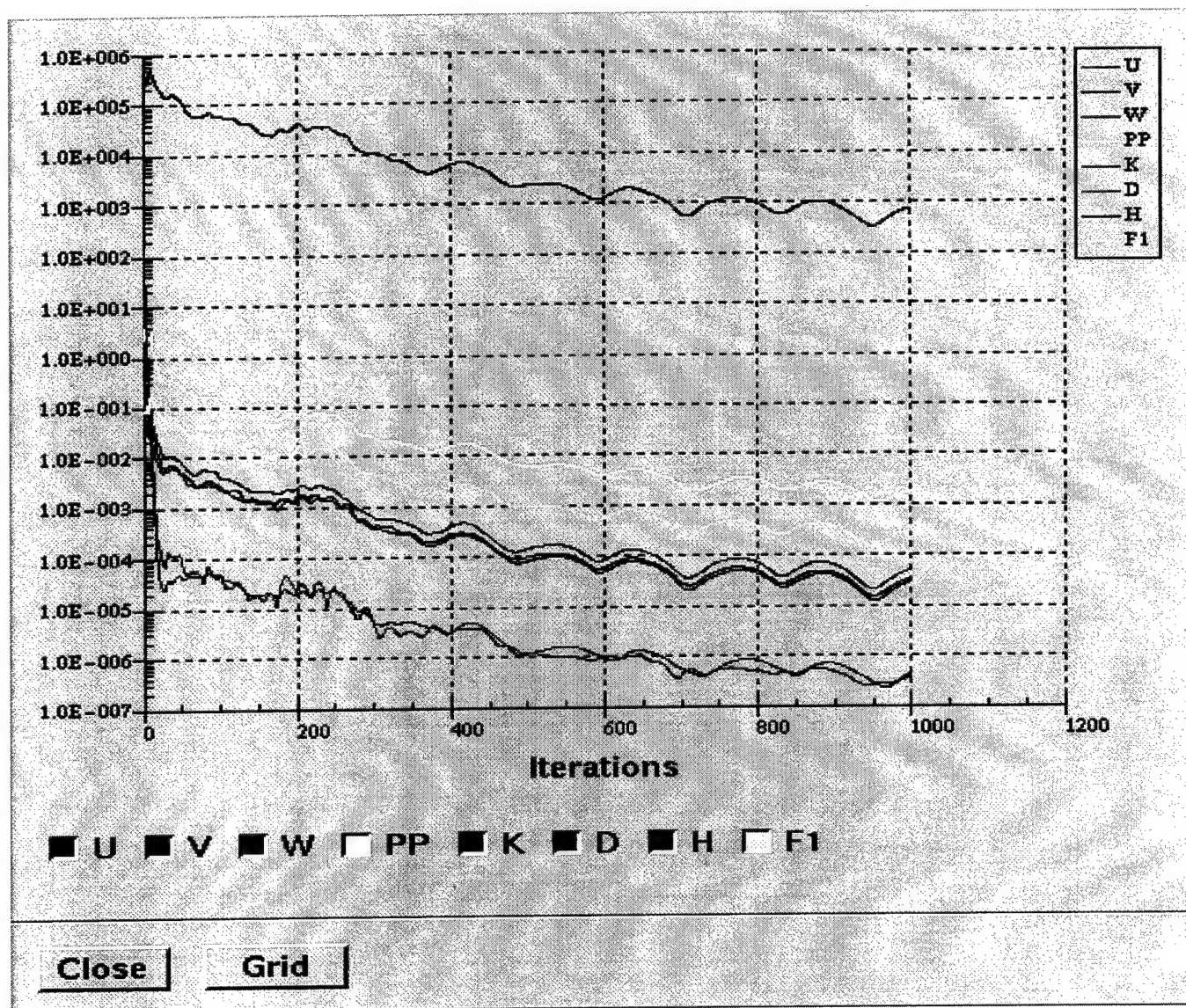**Figure 31. Isosurface of Smoke showing 16 percent concentration by volume.**

Figure 32. Residuals for Scenario C

THIS PAGE INTENTIONALLY LEFT BLANK

LIST OF REFERENCES

1. Levinson, J., Edwards, R., *Missile Inbound,*

2. Kay, D., Morris, M., *Fire Safety On Ship Developments into the 21st Century,* The Institute of Marine Engineers, 1994.

3. Tabarra, M., Kenrick, B., Matthews, R., *CFD Validation of Natural Smoke Movement in a Model Tunnel,* Centre for Tunnel Research and Development, 1996.

4. Chow, W., *Studies On The Stability Of Thermal Stratified Layer In a Forced Ventilation Fire Using Computational Fluid Dynamics,* Baywood Publishing Co, 1996.

5. Null, G., *Computer Simulation Of The Thermal Effects On A Concentric Canister Missile Launcher With a Fire In An Adjacent Compartment,* M.S. Thesis, Mechanical Engineering Department, Naval Postgraduate School, Sept, 1997.

6. Blay, D., Tuhault, J., Joubert, P., Modelling Smoke Movement in Corridors, Laboratoire D'Etudes des Systemes Thermiques et Energetiques.

7. CFD Research Corporation, CFD-ACE Theory Manual, Oct 1998.

8. CFD Research Corporation, CFD-GEOM, Oct 1998.

9. CFD Research Corporation, CFD-VIEW, Oct 1998.

10. CFD Research Corporation, CFD-GUI, Oct 1998.

THIS PAGE INTENTIONALLY LEFT BLANK

INITIAL DISTRIBUTION LIST

1. Defense Technical Information Center..2
8725 John J. Kingman Road, Ste 0944
Ft Belvoir, Virginia 22060-6218

2. Dudley Knox Library...2
Naval Postgraduate School
411 Dyer Road
Monterey, Ca 93943-5101

3. Engineering & Technology Curricular Office, Code 34.........................1
Naval Postgraduate School
Monterey, Ca 93943-5101

4. Department of Mechanical Engineering Code ME..............................1
Naval Postgraduate School
Monterey, Ca 93943-5101

5. Professor Matthew D. Kelleher..2
Mechanical Engineering Department, Code ME/KK
Naval Postgraduate School
Monterey, Ca 93943-5101

6. LT Mike Mehls, USN...1
18900 Pasnow Ave
Euclid Oh 44119

7. CFD Research Corporation..1
Cummings Research Park
215 Wynn Drive
Huntsville, Alabama 35805

8. NAWC-Weapons Division...1
Fire Research Office Code: 4B3100D
Attn: L. Bowman
China Lake, California 93555-6100

9. NAWC-Weapons Division...1
Fire Research Office Code: 4B3100D
Attn: Jim Hoover
China Lake, California 93555-6100

Attn: Norm Yarbrough
2531 Jefferson Davis Highway
Arlington, VA 22242-5160